



HEIDENHAIN

User's Manual

CNC PILOT 4290 B and Y Axis

NC Software
625 952-xx

English (en)
4/2010



CNC PILOT 4290 B and Y Axis

This manual describes functions and features that are available for the B axis, the Y axis and the tool magazine in the CNC PILOT 4290 with NC software number 625 952-xx (Release 7.1). This manual is a **supplement** to the CNC PILOT 4290 User's Manual.

1 B and Y Axis 7

1.1 Basics 8	
The Y axis 8	
The B axis 8	
The tool magazine 10	
1.2 Manual Control and Automatic Modes 11	
Automatic mode without reference run 11	
Magazine list 11	
Working with magazine tools 14	
Measuring and compensating magazine tools 18	
Tool compensation in automatic mode 19	
1.3 Programming Notes 20	
Milling contour position 20	
Cutting limit 20	
Drilling and milling in a tilted plane 21	
1.4 DIN PLUS: Section Codes 22	
PLATE MAGZN. section 22	
FRONT_Y, REAR_SIDE_Y section 22	
SURFACE_Y section 23	
1.5 DIN PLUS: Contours in the XY Plane 24	
Starting point of contour G170 Geo 24	
Linear element G171 Geo 24	
Circular arc G172/G173 Geo 25	
Hole G370 Geo 26	
Linear slot G371 Geo 26	
Circular slot G372/G373 Geo 27	
Full circle G374 Geo 27	
Rectangle G375 Geo 28	
Eccentric polygon G377 Geo 28	
Linear pattern in XY plane, G471-Geo 29	
Circular pattern in XY plane, G472 Geo 30	
Single surface G376 Geo 31	
Centric polygon G477 Geo 31	

1.6 DIN PLUS: Contours in the YZ Plane	32
Starting point of contour G180 Geo	32
Linear element G181 Geo	32
Circular arc G182/G183 Geo	33
Hole G380 Geo	34
Linear slot G381 Geo	34
Circular slot G382/G383 Geo	35
Full circle G384 Geo	35
Rectangle G385 Geo	36
Eccentric polygon G387 Geo	36
Linear pattern in YZ plane, G481-Geo	37
Circular pattern in YZ plane, G482-Geo	38
Single surface G386 Geo	39
Centric polygon G487 Geo	39
1.7 DIN PLUS: Working Planes	40
Tilting the working plane G16	41
1.8 DIN PLUS (Y Axis): Positioning Commands	42
Rapid traverse G0	42
Approach tool change point G14	42
Rapid traverse to machine coordinates G701	43
1.9 DIN PLUS: Magazine Tools	44
Insert magazine tool G714	44
Define tool position G712	47
Preselect tool G600	48
1.10 DIN PLUS: Linear and Circular Paths	49
Milling: Linear movement G1	49
Milling: Circular movement G2, G3—incremental center coordinates	50
Milling: Circular path G12, G13—absolute center coordinates	51
1.11 DIN PLUS (Y Axis): Milling Cycles	52
Area milling—roughing G841	52
Area milling—finishing G842	53
Centric polygon milling—roughing G843	54
Centric polygon milling—finishing G844	55
Pocket milling - roughing G845 (Y axis)	56
Pocket milling—finishing G846 (Y axis)	61
Engrave in XY plane G803	63
Engrave in YZ plane G804	64
Thread milling in XY plane G800	65
Thread milling in YZ plane G806	66
Hobbing G808	67
1.12 Simulation	68
Simulation of the tilted plane	68
Displaying the coordinate system	69
Position display with the B and Y axes	69

1.13 TURN PLUS: Tool Magazine and B Axis	70
Tool magazine	70
Tools for the B axis	70
1.14 TURN PLUS: Y Axis	71
Y axis - Basics	71
Definition of milling contours	72
1.15 TURN PLUS: XY Plane Contours	73
Reference data - XY front/XYR back	73
XY plane: Starting point of contour	74
XY plane: Linear element	75
XY plane: Arc	76
XY plane: Single hole	77
XY plane: Circle (full circle)	79
XY plane: Rectangle	80
XY plane: Polygon	81
XY plane: Linear slot	82
XY plane: Circular slot	83
XY plane: Linear drilling pattern	84
XY plane: Circular drilling pattern	85
XY plane: Linear figure pattern	86
XY plane: Circular figure pattern	87
XY plane: Single surface	88
XY plane: Centric polygon	88
1.16 TURN PLUS: YZ Plane Contours	89
Reference data - Y lateral surface	89
YZ plane: Starting point of contour	89
YZ plane: Linear element	90
YZ plane: Arc	91
YZ plane: Single hole	92
YZ plane: Circle (full circle)	94
YZ plane: Rectangle	95
YZ plane: Polygon	96
YZ plane: Linear slot	97
YZ plane: Circular slot	98
YZ plane: Linear drilling pattern	99
YZ plane: Circular drilling pattern	100
YZ plane: Linear figure pattern	101
YZ plane: Circular figure pattern	102
Single surface in YZ plane	103
Centric polygons in YZ plane	103
1.17 Example Programs	104
Machining with the Y axis	104
Machining with the B axis	109



1

B and Y Axis

1.1 Basics

The Y axis

With a Y axis you can drill and mill a workpiece on its front, back and lateral surfaces.

During use of the Y-axis, two axes interpolate linearly or circularly in the given working plane, while the third axis interpolates linearly. This enables you to machine slots or pockets, for example, with plane floors and perpendicular edges. By defining the spindle angle, you can determine the position of the milling contour on the workpiece.

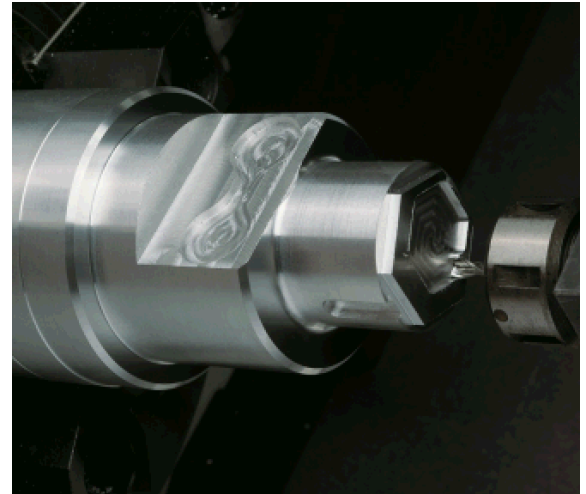
The CNC PILOT supports part program creation with the Y axis in:

- DIN PLUS
- TURN PLUS contour definition
- TURN PLUS working plan generation

The separation of contour description and machining also applies to milling with the Y axis. Contour regeneration is not available for milling operations.

Y axis contours are identified with section codes.

The **graphical simulation** shows the milling operation in the familiar lathe, front, and surface windows, as well as in the "side view (YZ)."



The B axis

Tilted working plane

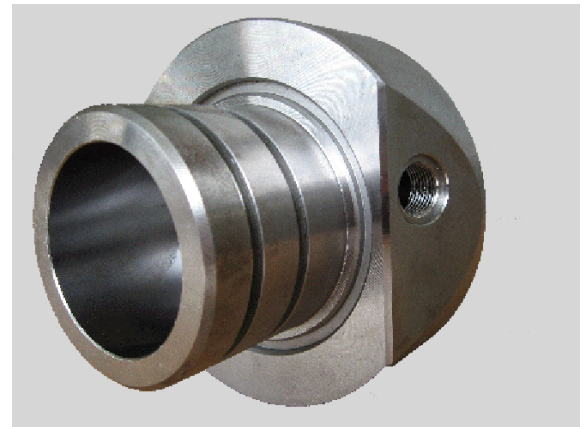
The B axis makes it possible to drill, bore and mill in oblique planes. To make programming easy, the coordinate system is tilted in such a way that you can define the drilling patterns and milling contours in the YZ plane. The actual drilling or milling operation is then performed in the tilted plane.

The separation of contour description and machining also applies to machining operations in tilted planes. Contour regeneration is not available.

Contours in tilted planes are identified with the section code SURFACE_Y.

The CNC PILOT supports part program creation with the B axis in DIN PLUS.

The **graphical simulation** shows the machining operation in a tilted working plane in the familiar lathe and front windows, as well as in the "side view (YZ)."



Tools for the B axis

Another advantage of the B axis is that it allows flexible use of the tools during turning operations. By tilting the B axis and rotating the tool you can bring it into positions that enable you to use one and the same tool to machine in the longitudinal and transverse (or radial and axial) directions on the main and opposing spindles.

In this way, you need fewer tools and fewer tool changes.

Tool data: All tools are described in the tool database by specifying the X, Z and Y dimensions as well as the compensation values. These dimensions are referenced to the **tilt angle B=0°**.

Another parameter that is maintained in the tool database is the **position angle**. It defines the working positions of tools that are not driven tools (turning tools).

The tilt angle of the B axis is not maintained with the tool data. This angle needs to be defined in the tool call or when inserting the tool.

Tool orientation and position display: For turning tools, the position the tool tip is calculated based on the orientation of the cutting edge. This orientation is **not** regenerated automatically when the B axis is tilted and/or rotated.

When the B axis has been moved manually, the control marks the position display **invalid**.

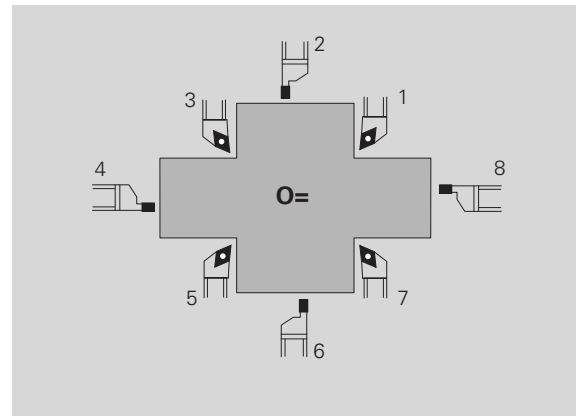
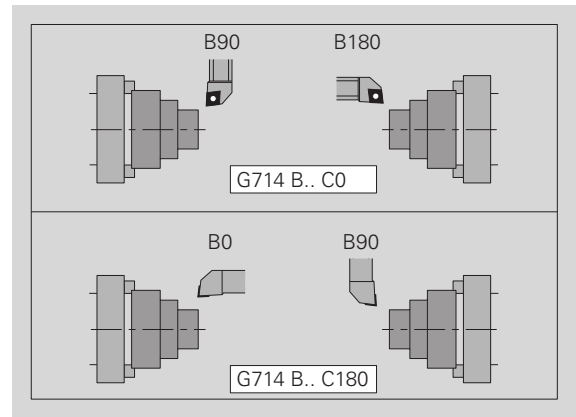
- Display with black digits: Position display is valid.
- Display with gray digits: Position display is invalid.

After moving the B axis, please check whether the orientation is still valid and reassign it, if necessary.

When orienting tools, the control distinguishes between roughing, finishing and button tools as well as recessing and threading tools (see figure).

- Tool positions 1, 3, 5 or 7 apply to roughing, finishing and button tools. The control recognizes neutral tools by the tool angle.
- Tool positions 2, 4, 6 or 8 apply to recessing and threading tools. Whether the tool is a "right-hand" or a "left-hand" tool is defined in the tool data.

Machine display: The **T** box in the machine display indicates the tool's pocket in the magazine. The current tilt angle of the B axis is taken into account in the compensation values shown in this box.



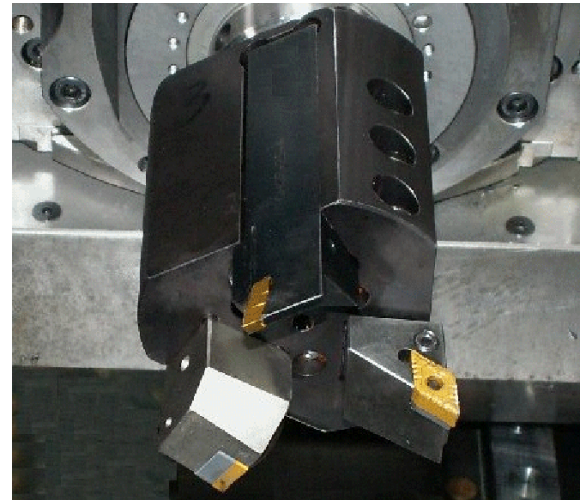
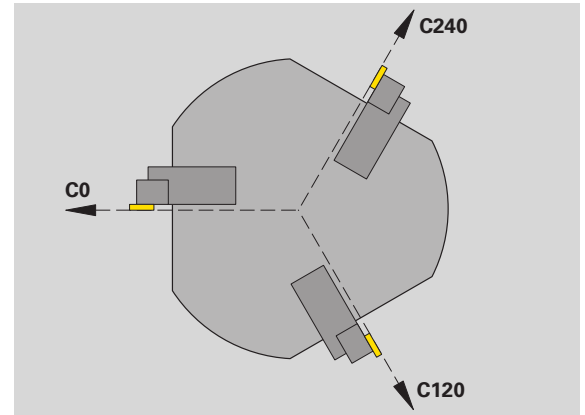
After tilting or rotating the B axis, the values given in the position display are invalid.

Multipoint tools for the B axis

If several tools are mounted on a tool holder, this is referred to as a "multipoint tool." Each cutting edge (tool) of a multipoint tool is assigned a separate ID number and description.

The **position angle**, which is identified by "C" in the figure, is included in the tool data. When a cutting edge (tool) of a multipoint tool is activated, the CNC PILOT will rotate the multipoint tool into the correct position. The position is determined from the position angle, to which the offset position angle from the tool change routine is added. This allows inserting the tool either in the "normal" attitude or "upside down."

The photo shows a multipoint tool with three cutting edges.



The tool magazine

The CNC PILOT supports a pocket-based tool magazine with up to 99 tools. Pocket-based means that each tool is assigned a specific pocket in the magazine. The machine operator assigns the pockets when setting up the magazine.

The magazine list indicates the current assignment of the tool magazine. The tools are entered in this list with their ID numbers.

Tool programming: The magazine tools are intended for the B axis. The command G714 is provided for changing and positioning the tools.

Alternatively, you can also use single commands (G0, G15, etc.) to program a tilting of the B axis and a rotation of the tool to the position angle. Please note, however, that you will need to declare the tool position with G712 in that case.

1.2 Manual Control and Automatic Modes

Automatic mode without reference run

As of software version 625 952-02:

You can start magazine programs and manual programs even if you have not traversed the reference marks in all the axes. To use this function, add a comment line to the program you want to start. In this comment line you define which axes are allowed without a reference status.

Syntax of the comment line:

- [**@0nn**]—where nn stands for the address letters of the non-referenced axes

Examples:

- [**@0B**]—the B axis does not have to be referenced
- [**@0BY**]—the B and Y axes do not have to be referenced



The functions for setting up the tool magazine and for inserting the magazine tools are interfaced to the CNC PILOT and the machine by the machine tool builder. The functionality provided on your machine may therefore differ from the functions described in this manual. Your machine manual provides more detailed information.

Magazine list

The magazine list indicates the current assignment of the tool magazine. When setting up the magazine list, you assign each tool a specific magazine pocket by entering the tool's ID number. For a **multipoint tool**, you can enter the ID number of any cutting edge of the tool. From that ID number, the CNC PILOT determines all other cutting edges of the multipoint tool, since all ID numbers of a multipoint tool are interlinked in the tool database.

The tool magazine can be set up in different ways:

- **Adding tools to the magazine by using the loading hatch:** see "Adding tools to the magazine by using the loading hatch" on page 12
- **Adding tools to the magazine from the machine working space:** see "Adding tools to the magazine from the machine working space" on page 13
- **Removing tools from the magazine:** see "Removing tools from the magazine" on page 13

The **tool life management** also applies to magazine tools without restriction.



Danger of collision

- Compare the magazine list with the tools actually in the tool magazine and check the tool data **before** executing the part program.
- The magazine list and the dimensions of the registered tools must correspond to the tools actually present, because the CNC PILOT used this data for all slide movements, protective zone monitoring, and other slide movements.

Adding tools to the magazine by using the loading hatch

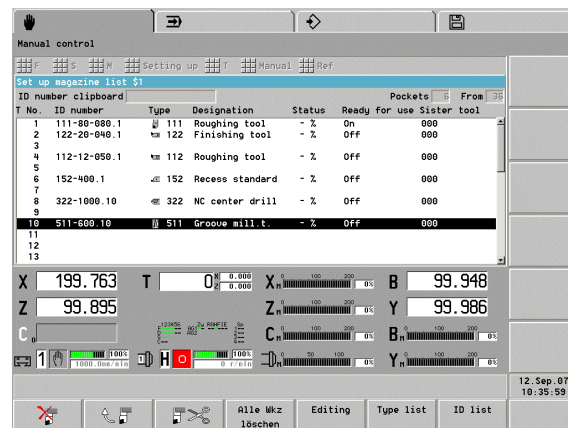
You can add a tool to the magazine by inserting it through the loading hatch and assigning the tool's ID number to a specific pocket of the magazine list.

To enter the tool ID number:

- ▶ Select "Setting up > Tool list > Setup list" in manual control mode.
- ▶ Place the cursor on the magazine pocket you want to assign to the tool.
- ▶ Select the tool's ID number from the database and confirm, or press the INS key and type in the ID number directly.
- ▶ Rotate the tool magazine to the corresponding position and insert the tool.



The functions "Compare tool list with NC program" and "Load tool list from NC program" are not available for the magazine list.



Adding tools to the magazine from the machine working space

Insert the tool in the tool holder and call the "Load plate" function. Enter the ID number of the tool and the magazine pocket number. The CNC PILOT inserts the tool in the magazine and enters the ID number in the magazine list.

- ▶ Insert the tool in the tool holder (in the machine's working space).
- ▶ Select "T > Magazine > Load plate" in manual control mode. The CNC PILOT opens the "Magazine: Load plate" dialog box.
- ▶ Enter the parameters and click OK to close the dialog box. The control loads the associated NC program.



- ▶ Activate the NC program with Cycle Start.

Parameters

- ID ID number of the magazine tool.
- P Pocket number in the tool magazine.
- B B axis angle. Angle to which the B axis is tilted.

The CNC PILOT

- inserts the tool in the magazine,
- enters the tool in the magazine list,
- moves the slide to the tool change position, and
- tilts the B axis,



Note on operation and display: This function is executed using an NC program. To activate the NC program, press Cycle Start.

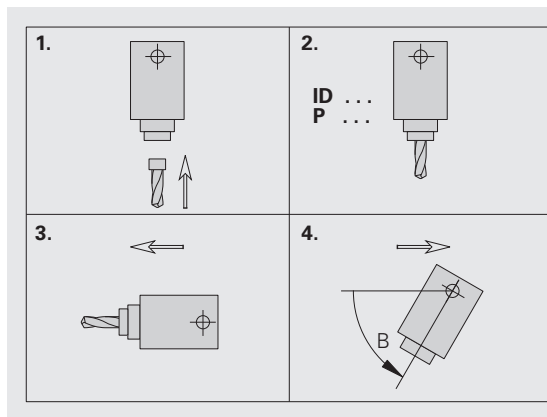
Removing tools from the magazine

Remove the tool from the magazine and delete the entry in the magazine list.

- ▶ Rotate the tool magazine to the corresponding position and remove the tool.
- ▶ Select "Setting up > Tool list > Setup list" in manual control mode.
- ▶ Place the cursor on the magazine pocket of the tool you removed.



- ▶ Press the soft key or the DEL key and click yes on the confirmation prompt. The control deletes the tool from the magazine list.



Working with magazine tools

Changing magazine tools

You can use this function to change the tool or to modify the tilt angle or position angle of the active tool.

- ▶ Select "T > Magazine > Tool change" in manual control mode. The control opens the "Magazine: Tool change" dialog box.
- ▶ Press the soft key, select the tool from the magazine list, enter the additional parameters and click OK to close the dialog box. The control loads the associated NC program.
- ▶ Activate the NC program with Cycle Start.

Magazine
list

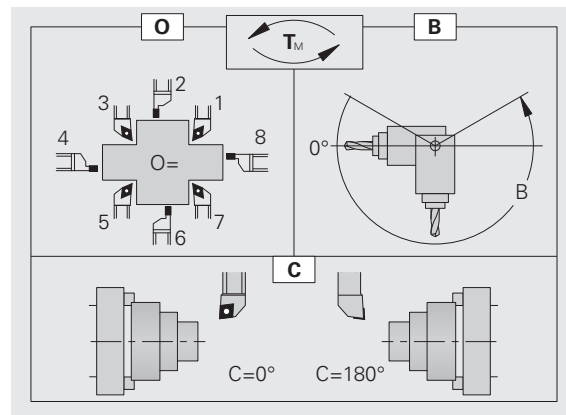


Parameters

- ID ID number of the magazine tool.
- O Orientation of turning tools. Position of the tool's cutting edge (see figure).
 - Tool positions 1, 3, 5, 7: For roughing, finishing and button tools (neutral tools are recognized by the tool angle).
 - Tool positions 2, 4, 6, 8: For recessing and threading cycles (a "right-hand" or "left-hand" tool is defined in the tool data).
- B B axis angle. Angle to which the B axis is tilted.
- C Offset position angle of turning tools.
 - 0°: Tool attitude "normal"
 - 180°: Tool attitude "upside down"
- H Shoe brake
 - 0: The brake is locked depending on the tool parameter (if "not driven" the brake is locked; if "driven" the brake is not locked)
 - 1: The brake is locked
 - 2: The brake is not locked

The CNC PILOT

- inserts the tool in the magazine,
- takes the specified tool from the magazine,
- moves to the tool change position,
- tilts the B axis,
- rotates the tool to the "normal" or "upside down" attitude (offset position angle C),
- calculates the tool data, taking the "orientation O," the B axis position and the position angle into account, and
- adjusts the brake settings.



Changing the tool position: If the call refers to the active tool, the slide moves to the tool change position and tilts the B axis or rotates the tool to the position angle.

Offset position angle: With the "offset position angle" you can position turning tools in the "normal" attitude or "upside down." When positioning the tool, the CNC PILOT also takes the basic setting saved in the tool database into account (position angle = position angle from the tool data + offset position angle).

Tool orientation: The CNC PILOT takes the orientation of the cutting edge into account when calculating the position of the tool tip. The control distinguishes between roughing, finishing and button tools as well as recessing and threading tools (see figure).



Note on operation and display: This function is executed using an NC program. To activate the NC program, press Cycle Start.

Declaring magazine tools

If there is a tool in the machine's working space when the control is switched off and on again, the tool needs to be redeclared. In the corresponding dialog box, the CNC PILOT automatically suggests the values in effect when the control was switched off.

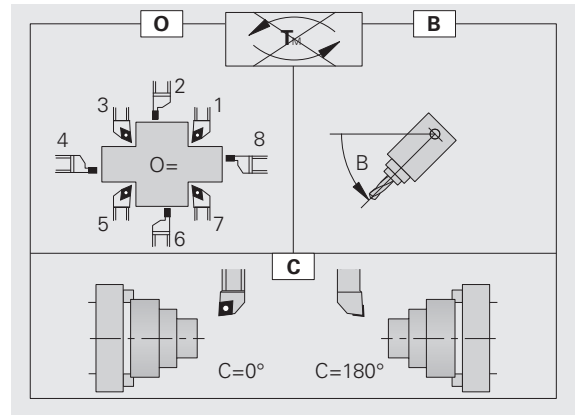
- ▶ Select "T > Magazine > Manual tool" in manual control mode. The control opens the "Magazine: Manual tool" dialog box.
- ▶ Press the soft key, enter the B axis angle, check all other parameters and click OK to close the dialog box. The control loads the associated NC program.
- ▶ Activate the NC program with Cycle Start.

Magazine
list



Parameters

- ID ID number of the magazine tool.
- P Pocket number in the tool magazine.
- O Orientation of turning tools. Position of the tool's cutting edge (see figure).
- Tool positions 1, 3, 5, 7: For roughing, finishing and button tools (neutral tools are recognized by the tool angle).
 - Tool positions 2, 4, 6, 8: For recessing and threading cycles (a "right-hand" or "left-hand" tool is defined in the tool data).
- B B axis angle. Angle to which the B axis is tilted.
- C Offset position angle of turning tools.
- 0°: Tool attitude "normal"
 - 180°: Tool attitude "upside down"



Parameters

H Shoe brake

- 0: The brake is locked depending on the tool parameter (if "not driven" the brake is locked; if "driven" the brake is not locked)
- 1: The brake is locked
- 2: The brake is not locked

The CNC PILOT

- moves to the tool change position,
- tilts the B axis,
- rotates the tool to the "normal" or "upside down" attitude (offset position angle C),
- calculates the tool data, taking the "orientation O," the B axis position and the position angle into account, and
- adjusts the brake settings.



- The information on the tool in the tool holder is not saved when the control is switched off. HEIDENHAIN therefore recommends to remove magazine tools from the working space before switching off the control.
- Note on operation and display: This function is executed using an NC program. To activate the NC program, press Cycle Start.

Returning tools to the magazine

The "Return tool to magazine" function moves the tool from the machine's working space back into the magazine. The tool carrier then approaches the tool change position and tilts the B axis to the specified angle.

- ▶ Select "T > Magazine > Return tool" in manual control mode. The control opens the "Magazine: Return tool" dialog box.
- ▶ Enter the "B axis angle B" parameter and click OK to close the dialog box. The control loads the associated NC program.

- ▶ Activate the NC program with Cycle Start.



Parameters

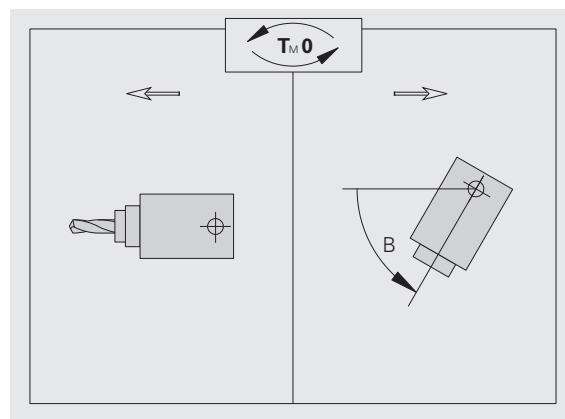
B B axis angle. Angle to which the B axis is tilted.

The CNC PILOT

- inserts the tool in the magazine,
- moves to the tool change position,
- tilts the B axis,



- Note on operation and display: This function is executed using an NC program. To activate the NC program, press Cycle Start.



Tilting the B axis in manual control mode

You can either use the tool change call for positioning the B axis or you can tilt the axis manually with the handwheel or PLC keys.

Tool change call: When you call the tool change function, the entries default to the current values. Specify the required B axis angle and activate the function.

Manual tilting: The B axis is tilted by using the handwheel. You can also move the B axis with the PLC keys if your control has been specially prepared for this functionality by the machine tool builder. Your machine manual provides more detailed information.

When you tilt the B axis manually, the new B axis angle is taken into account, but a change in the tool orientation is not recognized. The control therefore marks the actual position displays for X and Z **invalid** (gray digits). In the next tool call, the CNC PILOT newly calculates the position of the tool tip and marks the position displays for X and Z valid.



Please note that the position displays for X and Z (machine display) will display invalid values as soon as the B axis is tilted manually. The CNC PILOT indicates this by displaying the position values in gray.

Measuring and compensating magazine tools

Measuring tools: The function determines the tool lengths referenced to the current tilt angle of the B axis and the position angle of the tool. These are the values that are indicated on the display. The control additionally converts the measured data into dimensions referenced to the position B=0 and saves them in the tool database.

- ▶ Select "Setting up > Tool set-up > Tool measuring" in manual control mode. The control indicates the current measurement values in the "Tool measuring T..." dialog box.
- ▶ Measure and enter the tool dimensions and click OK to close the dialog box.

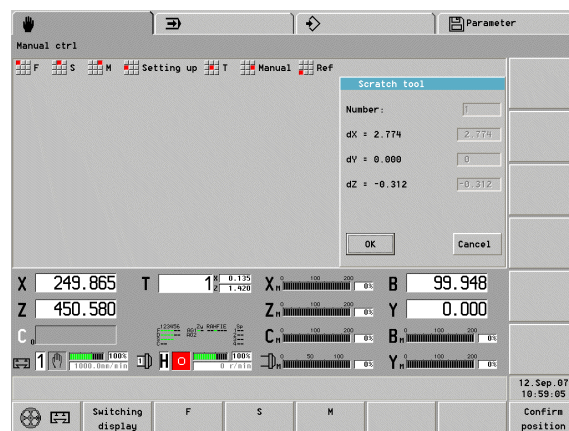
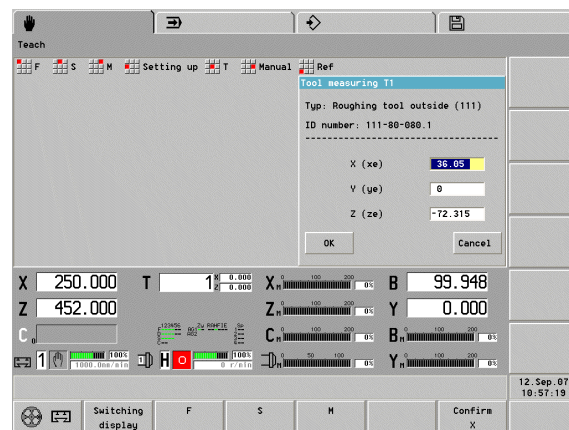
The control

- deletes the compensation values and
- enters the tool dimensions in the database.

Determining compensation values: The compensation values are determined and displayed referenced to the current tilt angle of the B axis and the position angle of the tool. The control converts the measured data into dimensions referenced to the position B=0 and saves them in the tool database.

- ▶ Select "Setting up > Tool set-up > Tool compensation" in manual control mode. In the "Scratch tool" dialog box, the control indicates the current compensation values referenced to position B=0.
- ▶ Determine the compensation values and click OK to close the dialog box.

The control applies the compensation values.



Tool compensation in automatic mode

Tool compensation: Determine the compensation values referenced to the current tilt angle of the B axis and the position angle of the tool. The control converts the measured data into dimensions referenced to the position B=0 and saves them in the tool database.

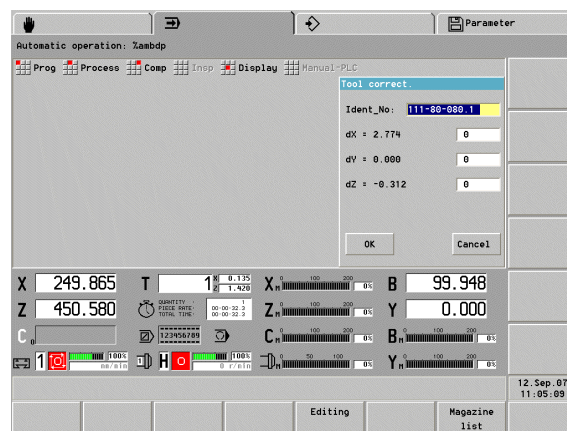
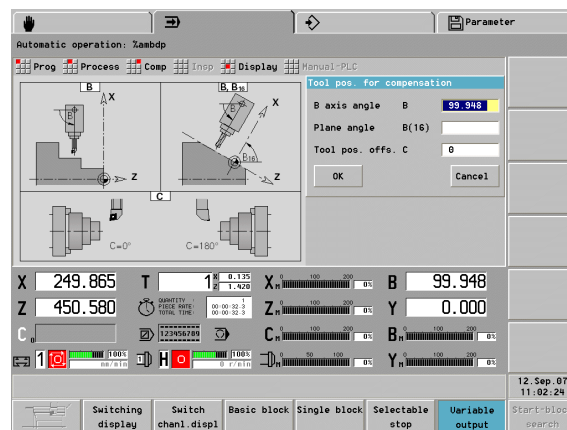
- ▶ Select “Comp > Tool compensation” in automatic mode. The control opens the “Tool correct.” dialog box.
- ▶ Enter the parameters and click OK to close the dialog box.
- ▶ In the “Tool correct.” dialog box, the control indicates the compensation values referenced to the B axis angle specified in the previous dialog box.
- ▶ Enter the new compensation values.

In the “T” box (machine display), the control indicates the compensation values referenced to the current B axis angle and the tool position angle.



- The CNC PILOT saves the tool compensation data in the tool database, together with the other tool data.
- If the B axis is tilted, the CNC PILOT takes the tool compensation data into account when calculating the tool tip position.

Additive compensation values are independent of the tool data. The compensation values are effective in the X, Y and Z directions. Tilting the B axis has no influence on additive compensation values.



1.3 Programming Notes

Milling contour position

Define the reference plane or the reference diameter in the section code. Specify the depth and position of a milling contour (pocket, island) in the contour definition:

- With **depth P** programmed in the previous G308 cycle.
- Alternatively on figures: Cycle parameter **depth P**.

The **algebraic sign of "P"** defines the position of the milling contour:

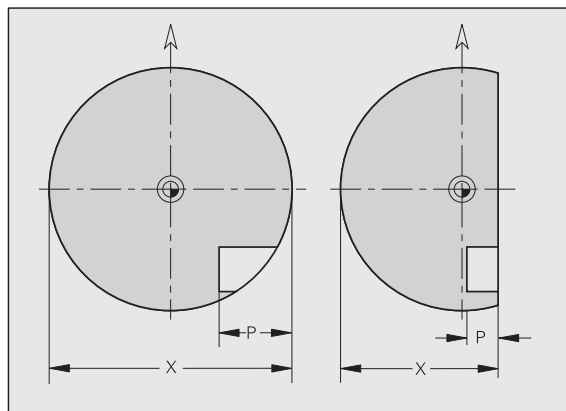
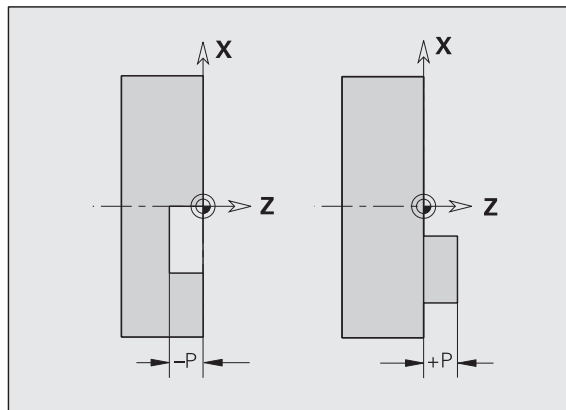
- $P < 0$: Pocket
- $P > 0$: Island

Milling contour position			
Section	P	Surface	Milling floor
STIRN [FRONT]	$P < 0$	Z	$Z + P$
	$P > 0$	$Z + P$	Z
RUECKSEITE [REAR SIDE]	$P < 0$	Z	$Z - P$
	$P > 0$	$Z - P$	Z
MANTEL [SURFACE]	$P < 0$	X	$X + (P * 2)$
	$P > 0$	$X + (P * 2)$	X

- X: Reference diameter from the section code
- Z: Reference plane from the section code
- P: Depth from G308 or from the figure definition



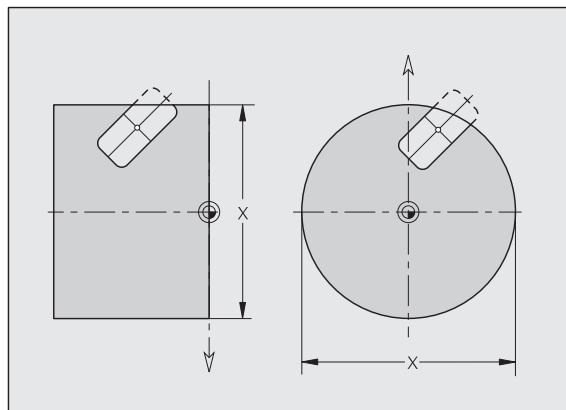
The area milling cycles mill the surface specified in the contour definition. **Islands** within this surface are not taken into consideration.



Cutting limit

If parts of the milling contour lie outside of the turning contour, you must limit the machining area with the **area diameter X / reference diameter X** (parameters of the section code or of the figure definition).

The cutting limits are also effective when milling in a tilted plane.



Drilling and milling in a tilted plane

HEIDENHAIN recommends tilting the coordinate system in such a way that you can define the drilling patterns and milling contours in the YZ plane. This has the advantage that you can then use all the contour, figure and pattern definitions for the YZ plane.

The drilling and milling cycles themselves are executed in the tilted plane. These cycles determine the position of the tilted plane from the contour definitions.

It is also a good idea to tilt the B axis with G714 because this G function includes the calculation of the tool position.

The following programming sequence is thus recommended:

- Rotate and shift the coordinate system for the tilted plane with the section code SURFACE_Y (see “SURFACE_Y section” on page 23).
- Define the drilling patterns and milling contours in the YZ plane.
- Position the B axis with G714.
- Activate the YZ plane with G19.
- Use the drilling and milling cycles for machining.

Alternatively, you can tilt the working plane with G16 and then execute the machining operations in the tilted plane.

Please note that the tool orientation is not regenerated automatically when you position the B axis with the single commands G0 or G15. Program G712 to have the tool position recalculated.

1.4 DIN PLUS: Section Codes

For lathes equipped with a tool magazine and/or a Y axis, the following section codes are available.

PLATE MAGZN. section

In the PLATE MAGZN. section, you list all the tools that are used in the NC program. This list is used when programming G714 (insert magazine tool). The entries can be made in any order.

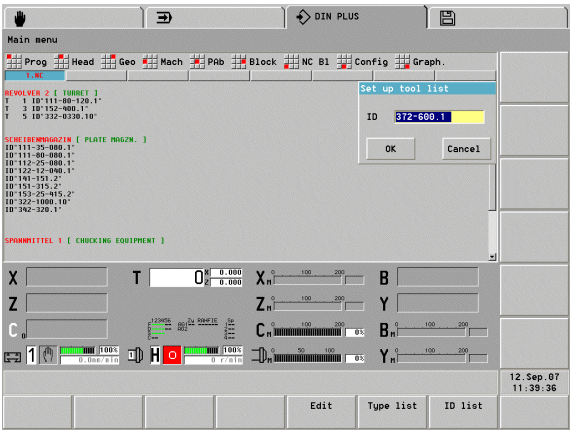
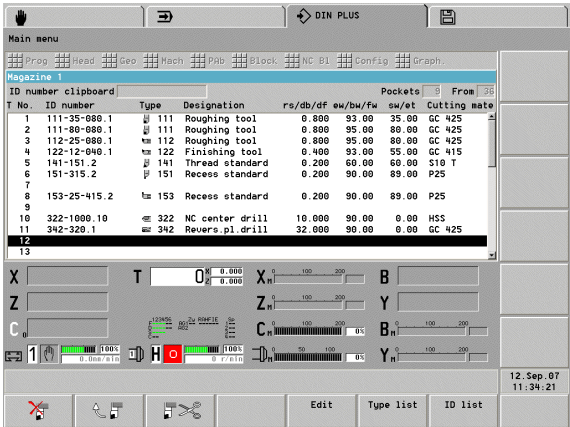
To create/edit the list of magazine tools:

- ▶ Select “Head > Set up tool list.”
- ▶ Select the tools from the database and enter them in the list.
- ▶ Press the ESC key to conclude the list.



To enter or edit individual magazine tools:

- ▶ Position the cursor in the PLATE MAGZN. section.
- ▶ To enter a new tool: Press the INS key.
- ▶ To edit a tool: Press RETURN or double-click with the left mouse button.
- ▶ Edit the “Set up tool list” dialog box.



FRONT_Y, REAR_SIDE_Y section

The section code identifies the XY plane (G17) and the reference plane of the contour (Z direction).

Parameters

- X Area diameter (as cutting limit)
- Z Position of the reference plane—default: 0
- C Spindle position—default: 0

SURFACE_Y section

The section code identifies the YZ plane (G19). For machines equipped with a B axis, it defines the tilted plane.

Without B axis: The reference diameter defines the contour position in the X direction; the C axis angle defines the position on the workpiece.

Parameters

- X Reference diameter
- C C axis angle—Defines the spindle position

With B axis (see figures): SURFACE_Y additionally performs the following transformations and rotations for the tilted plane:

- Shifts the coordinate system to the position I, K
- Rotates the coordinate system by the angle B; reference point: I, K
- H=0: Shifts the rotated coordinate system by $-I$. The coordinate system is moved "back."

Parameters

- X Reference diameter
- C C axis angle—Defines the spindle position
- B Plane angle: Positive Z axis
- I Plane reference in X direction (radius)
- K Plane reference in Z direction
- H Automatic shift of the coordinate system (default: 0)
 - 0: The rotated coordinate system is shifted by $-I$
 - 1: The coordinate system is not shifted

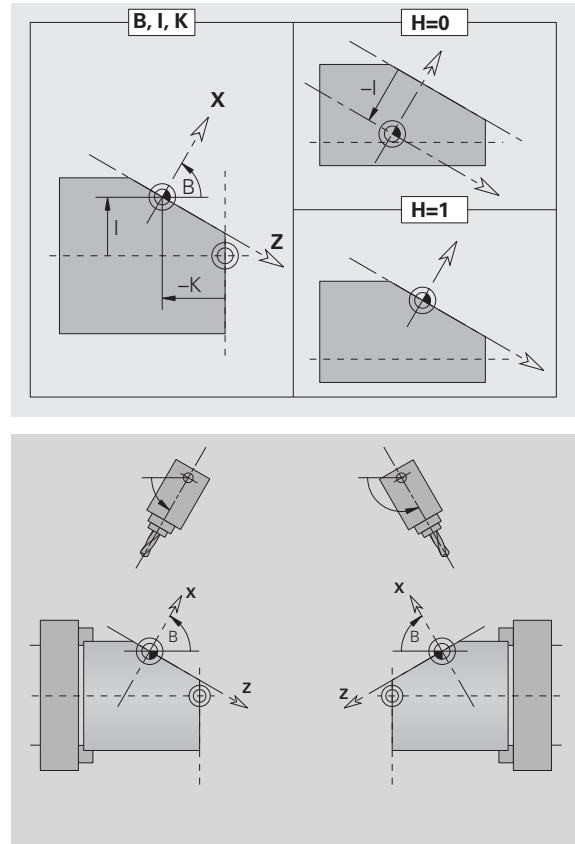
Shifting "back" coordinate system: The CNC PILOT evaluates the reference diameter for the cutting limit. This value is also used as the reference value for the depth that you program for drilling operations and milling contours.

Since the reference diameter is referenced to the current zero point, it is recommended when working in a tilted plane, to shift the rotated coordinate system "back" by the distance $-I$. If the cutting limits are not needed, for example for drilling holes, you can disable the shift of the coordinate system (H=1) and set the reference diameter to 0.



Please note:

- X is the infeed axis in a tilted coordinate system. X coordinates are entered as diameter coordinates.
- Mirroring the coordinate system has no effect on the reference axis of the tilt angle ("B axis angle" of G714).



Example: "SURFACE_Y"

```
PROGRAMMKOPF [PROGRAM HEAD]
...
CONTOUR Q1 X0 Z600
ROHTEIL [BLANK]
...
FERTIGTEIL [FINISHED PART]
...
MANTEL_Y X118 C0 B130 I59 K0 [SURFACE_Y]
...
BEARBEITUNG [MACHINING]
...
```

1.5 DIN PLUS: Contours in the XY Plane

Starting point of contour G170 Geo

G170 defines the starting point of a contour in the XY plane.

Parameters

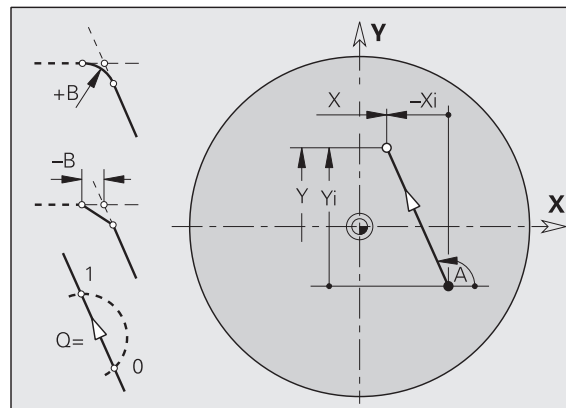
- X Starting point of contour (radius)
Y Starting point of contour

Linear element G171 Geo

G171 defines a line segment in a contour of the XY plane.

Parameters

- X End point (radius)
Y End point
A Angle to positive X axis
B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
- No entry: Tangential transition
 - B=0: No tangential transition
 - B>0: Rounding radius
 - B<0: Chamfer width
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
- Q=0: Near point of intersection
 - Q=1: Far point of intersection



Programming X, Y: Absolute, incremental, modal or “?”

Circular arc G172/G173 Geo

G172/G173 defines a circular arc in a contour of the XY plane. Direction of rotation: See help graphic

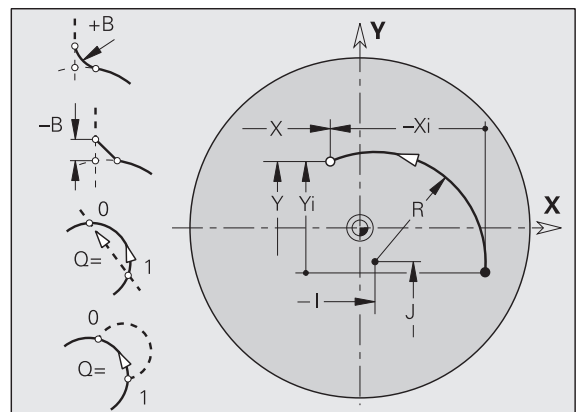
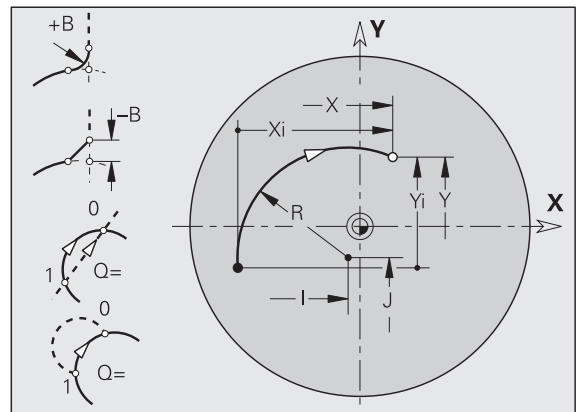
Parameters

- X End point (radius)
- Y End point
- I Center in X direction (radius)
- J Center in Y direction
- R Radius
- B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
 - No entry: Tangential transition
 - B=0: No tangential transition
 - B>0: Rounding radius
 - B<0: Chamfer width
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
 - For a transition to a line segment:
 - Q=0: Near point of intersection
 - Q=1: Far point of intersection
 - For a transition to a circular arc:
 - Q=0: Far point of intersection
 - Q=1: Near point of intersection



Programming

- **X, Y:** Absolute, incremental, modal or "?"
- **I, J:** Absolute or incremental
- End point must not be the starting point (no full circle).

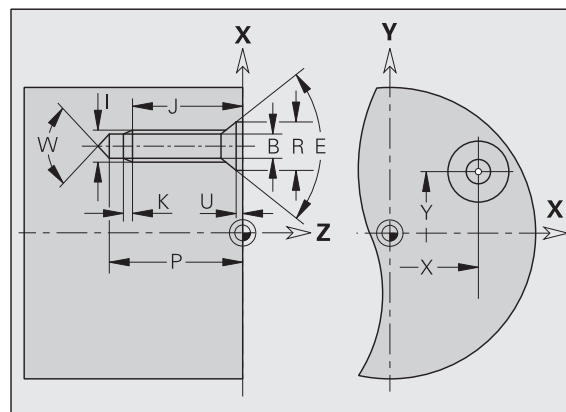


Hole G370 Geo

G370 defines a hole with countersinking and thread in the XY plane.

Parameters

- X Center of hole (radius)
- Y Center of hole
- B Hole diameter
- P Depth of hole (excluding point)
- W Point angle (default: 180°)
- R Sinking diameter
- U Sinking depth
- E Sinking angle
- I Thread diameter
- J Thread depth
- K Thread runout length
- F Thread pitch
- V Left-hand or right-hand thread (default: 0)
 - V=0: Right-hand thread
 - V=1: Left-hand thread
- A Angle to Z axis – inclination of the hole
 - Front face (range: $-90^\circ < A < 90^\circ$)—default: 0°
 - Rear side (range: $90^\circ < A < 270^\circ$)—default: 180°
- O Centering diameter

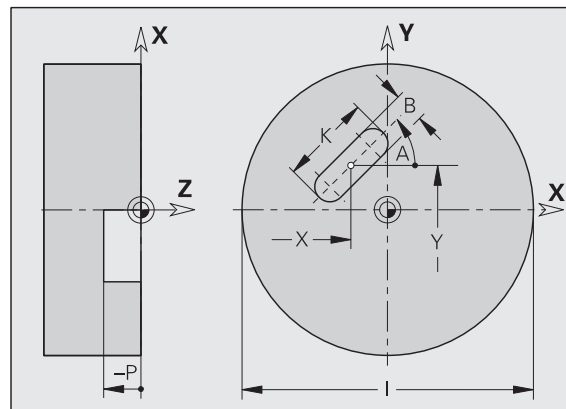


Linear slot G371 Geo

G371 defines the contour of a linear slot in the XY plane.

Parameters

- X Center of slot (radius)
- Y Center of slot
- K Slot length
- B Slot width
- A Angle of slot length (reference: positive X axis)—default: 0°
- P Depth/height (default: "P" from G308)
 - P<0: Pocket
 - P>0: Island
- I Area diameter (as cutting limit)
 - No entry: "X" from section code
 - "I" overwrites "X" from section code



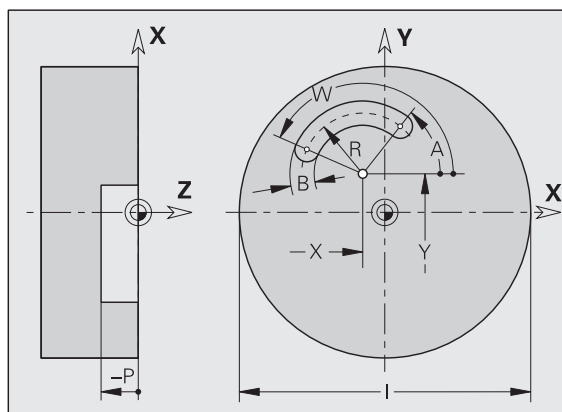
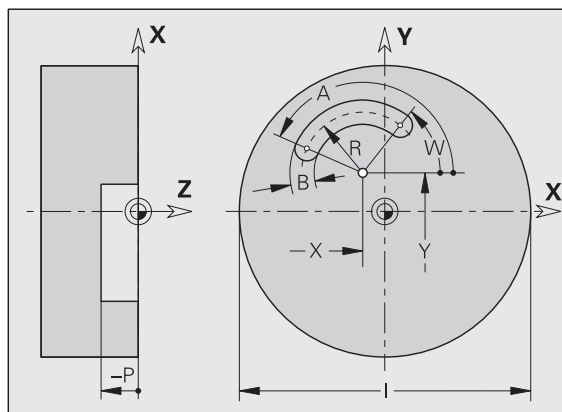
Circular slot G372/G373 Geo

G372/G373 defines a circular slot in the XY plane.

- G372: Circular slot clockwise
- G373: Circular slot counterclockwise

Parameters

- X Center of slot curvature (radius)
- Y Center of slot curvature
- R Curvature radius (reference: center point path of the slot)
- A Starting angle; reference: positive X axis (default: 0°)
- W End angle; reference: positive X axis (default: 0°)
- B Slot width
- P Depth/height (default: "P" from G308)
 - $P < 0$: Pocket
 - $P > 0$: Island
- I Area diameter (as cutting limit)
 - No entry: "X" from section code
 - "I" overwrites "X" from section code

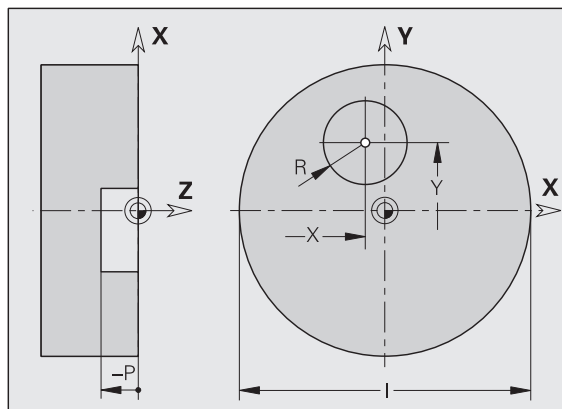


Full circle G374 Geo

G374 defines a full circle in the XY plane.

Parameters

- X Circle center (radius)
- Y Circle center
- R Circle radius
- P Depth/height (default: "P" from G308)
 - $P < 0$: Pocket
 - $P > 0$: Island
- I Area diameter (as cutting limit)
 - No entry: "X" from section code
 - "I" overwrites "X" from section code

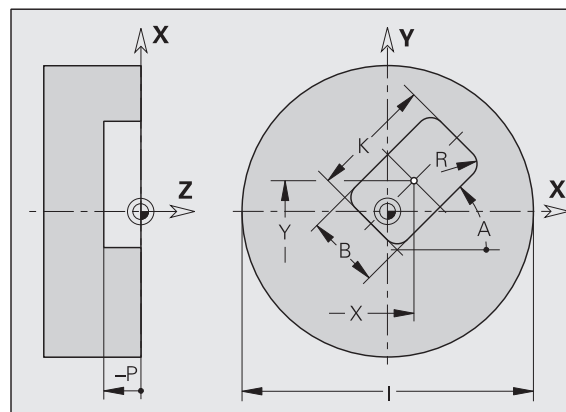


Rectangle G375 Geo

G375 defines a rectangle in the XY plane.

Parameters

- X Center of rectangle (radius)
- Y Center of rectangle
- K Length of rectangle
- B (Height) width of rectangle
- R Chamfer/rounding (default: 0)
 - $R > 0$: Radius of rounding arc
 - $R < 0$: Chamfer width
- A Angle to X axis (default: 0°)
- P Depth/height (default: "P" from G308)
 - $P < 0$: Pocket
 - $P > 0$: Island
- I Area diameter (as cutting limit)
 - No entry: "X" from section code
 - "I" overwrites "X" from section code

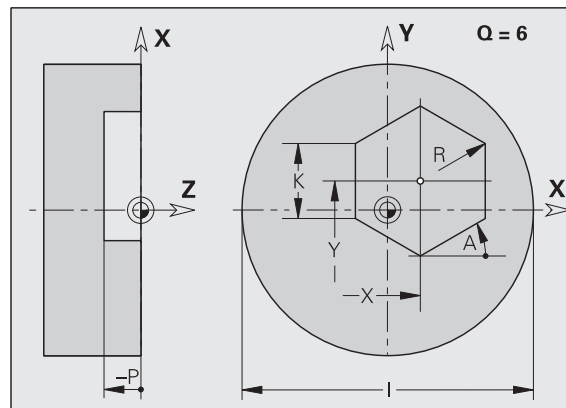


Eccentric polygon G377 Geo

G377 defines the contour of an eccentric polygon in the XY plane.

Parameters

- X Center point of polygon (radius)
- Y Center point of polygon
- Q Number of edges ($Q \geq 3$)
- A Angle to X axis (default: 0°)
- K Edge length
 - $K > 0$: Edge length
 - $K < 0$: Inside diameter
- R Chamfer/rounding—default: 0
 - $R > 0$: Radius of rounding arc
 - $R < 0$: Chamfer width
- P Depth/height (default: "P" from G308)
 - $P < 0$: Pocket
 - $P > 0$: Island
- I Area diameter (as cutting limit)
 - No entry: "X" from section code
 - "I" overwrites "X" from section code

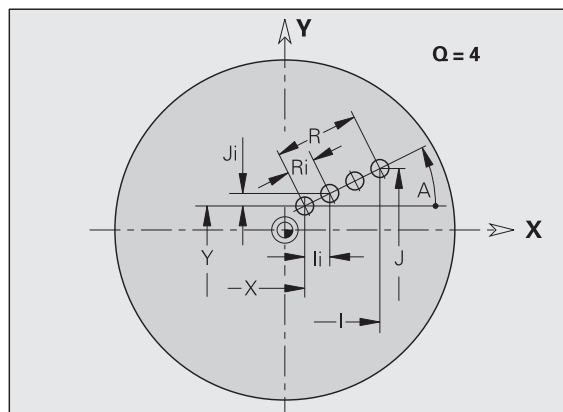


Linear pattern in XY plane, G471-Geo

G471 defines a linear pattern in the XY plane. G471 affects the hole or figure defined in the following block (G370 to G375, G377).

Parameters

Q	Number of figures
X	Starting point of pattern (radius)
Y	Starting point of pattern
I	End point of pattern (X direction; radius)
J	End point of pattern (Y direction)
li	Distance in X direction between two figures
Ji	Distance in Y direction between two figures
A	Angle of longitudinal axis to X axis
R	Total length of pattern
Ri	Distance between two figures (pattern distance)



Programming notes

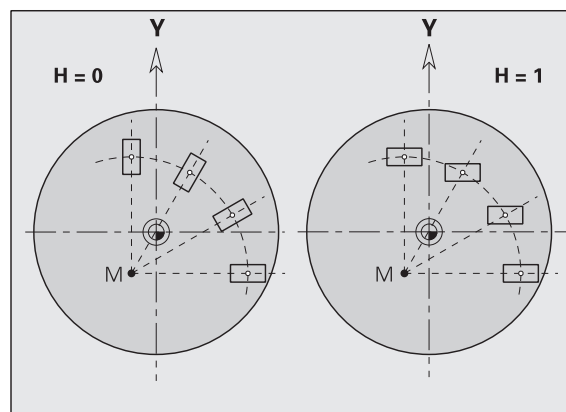
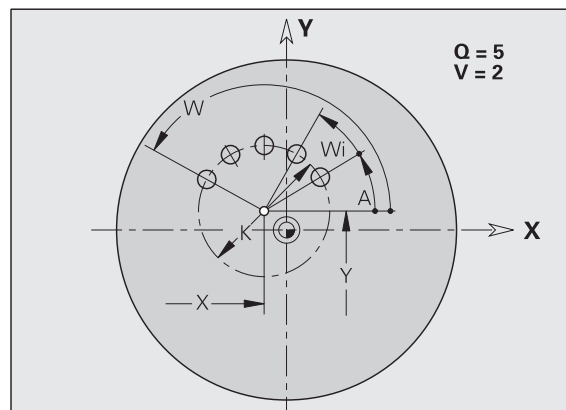
- Program the hole/figure in the following block without a center.
- The milling cycle (MACHINING section) calls the hole/figure in the following block - not the pattern definition.

Circular pattern in XY plane, G472 Geo

G472 defines a circular pattern in the XY plane. G472 is effective for the figure defined in the following block (G370 to G375, G377).

Parameters

- Q Number of figures
- K Pattern diameter
- A Starting angle—position of the first figure; reference: positive X axis (default: 0°)
- W End angle—position of the last figure; reference: positive X axis; (default: 360°)
- Wi Angle between two figures
- V Direction—orientation (default: 0)
 - V=0, without W: Figures are arranged on a full circle
 - V=0, with W: Figures are arranged on the longer circular arc
 - V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
 - V=1, with W: Clockwise
 - V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
 - V=2, with W: Counterclockwise
 - V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)
- X Center of pattern (radius)
- Y Center of pattern
- H Position of the figures (default: 0)
 - H=0: Normal position; the figures are rotated about the circle center (rotation)
 - H=1: Original position; the position of the figures relative to the coordinate system remains unchanged (translation)



- Program the hole/figure in the following block without a center. Exception: **circular slot**.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.

Single surface G376 Geo

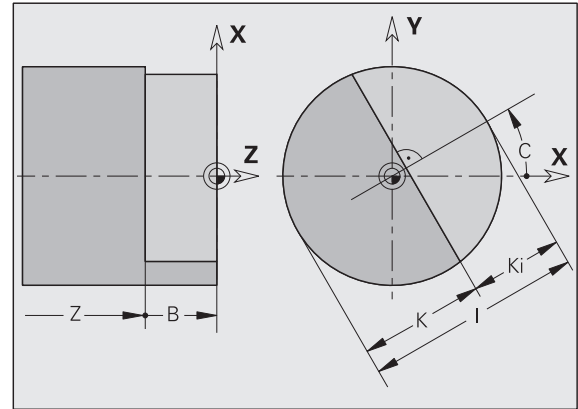
G376 defines a surface in the XY plane.

Parameters

- Z Reference edge (default: "Z" from section code)
- K Residual depth
- Ki Depth
- B Width (reference: reference edge Z)
- $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction
- I Surface diameter
- No entry: "X" from section code
 - "I" overwrites "X" from section code
- C Angular position of surface (default: "C" from section code)



Whether the surface lies on the front face or rear side has no effect on the evaluation of the algebraic sign for "width B."



Centric polygon G477 Geo

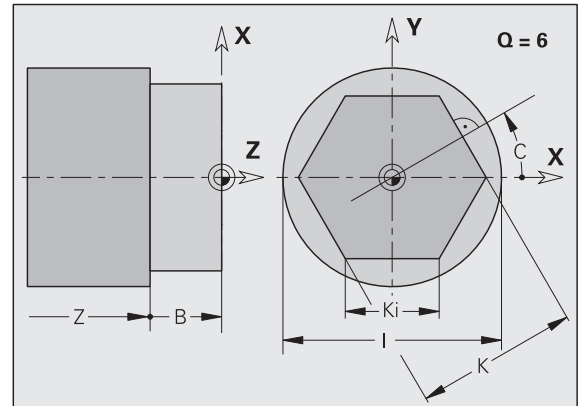
G477 defines polygonal surfaces in the XY plane.

Parameters

- Z Reference edge (default: "Z" from section code)
- K Inside diameter (width across flats)
- Ki Length of side
- B Width (reference: reference edge Z)
- $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction
- Q Number of sides ($Q \geq 2$)
- I Surface diameter
- No entry: "X" from section code
 - "I" overwrites "X" from section code
- C Angular position of surface (default: "C" from section code)



Whether the surface lies on the front face or rear side has no effect on the evaluation of the algebraic sign for "width B."



1.6 DIN PLUS: Contours in the YZ Plane

Starting point of contour G180 Geo

G180 defines the starting point of a contour in the YZ plane.

Parameters

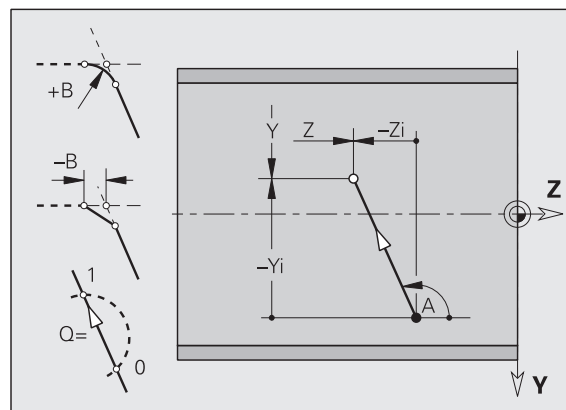
- Y Starting point of contour
- Z Starting point of contour

Linear element G181 Geo

G181 defines a line segment in a contour of the YZ plane.

Parameters

- Y End point
- Z End point
- A Angle to positive Z axis
- B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
 - No entry: Tangential transition
 - B=0: No tangential transition
 - B>0: Rounding radius
 - B<0: Chamfer width
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
 - Q=0: Near point of intersection
 - Q=1: Far point of intersection



Programming Y, Z: Absolute, incremental, modal or “?”

Circular arc G182/G183 Geo

G182/G183 defines a circular arc in a contour of the YZ plane. Direction of rotation: See help graphic

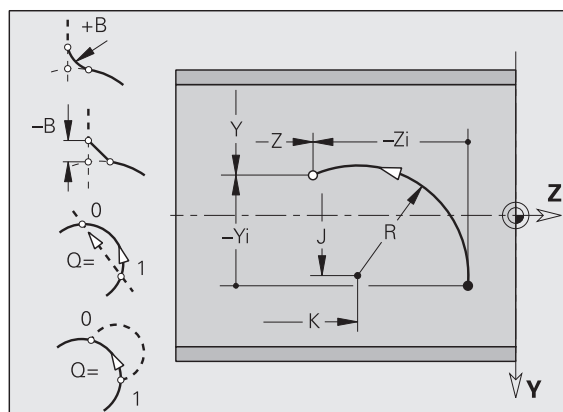
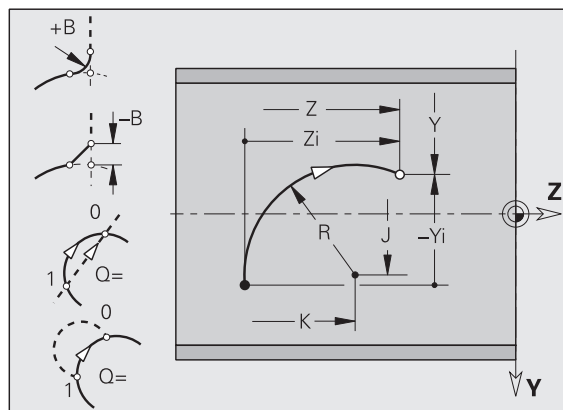
Parameters

- Y End point (radius)
- Z End point
- J Center (Y direction)
- K Center (Z direction)
- R Radius
- B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
 - No entry: Tangential transition
 - B=0: No tangential transition
 - B>0: Rounding radius
 - B<0: Chamfer width
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
 - For a transition to a line segment:
 - Q=0: Near point of intersection
 - Q=1: Far point of intersection
 - For a transition to a circular arc:
 - Q=0: Far point of intersection
 - Q=1: Near point of intersection



Programming

- **Y, Z:** Absolute, incremental, modal or "?"
- **J, K:** Absolute or incremental
- End point must not be the starting point (no full circle).

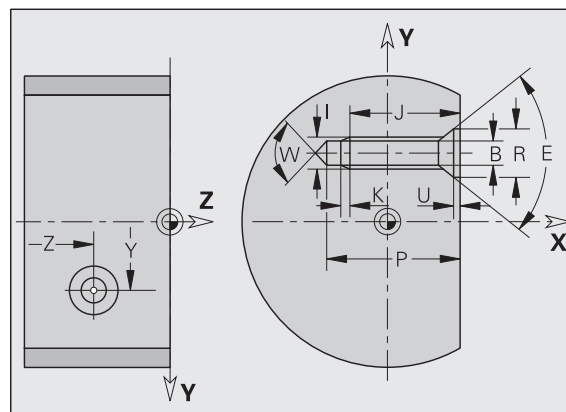


Hole G380 Geo

G380 defines a single hole with countersinking and thread in the YZ plane.

Parameters

- Y Center of hole
- Z Center of hole
- B Diameter of hole
- P Depth of hole (excluding point)
- W Point angle (default: 180°)
- R Sinking diameter
- U Sinking depth
- E Sinking angle
- I Thread diameter
- J Thread depth
- K Start of thread (runout length)
- F Thread pitch
- V Left-hand or right-hand thread (default: 0)
 - V=0: Right-hand thread
 - V=1: Left-hand thread
- A Angle to X axis; range: $-90^\circ < A < 90^\circ$
- O Centering diameter

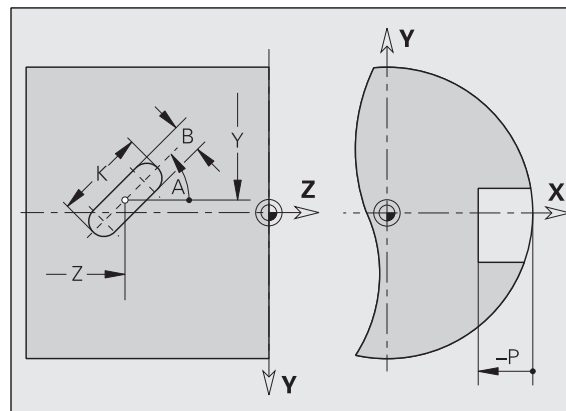


Linear slot G381 Geo

G381 defines the contour of a linear slot in the YZ plane.

Parameters

- Y Center of slot
- Z Center of slot
- X Reference diameter
 - No entry: "X" from section code
 - "X" from G381 overwrites "X" from section code
- A Angle to Z axis (default: 0°)
- K Slot length
- B Slot width
- P Pocket depth (default: "P" from G308)



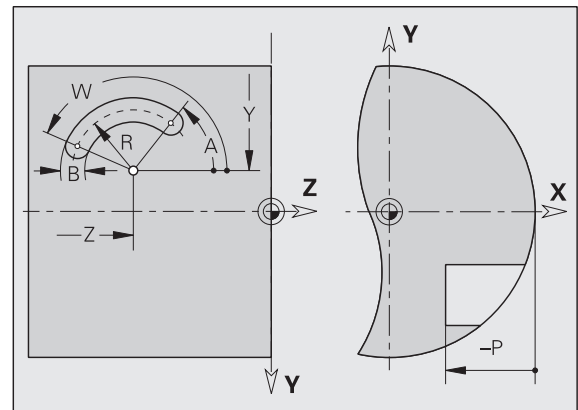
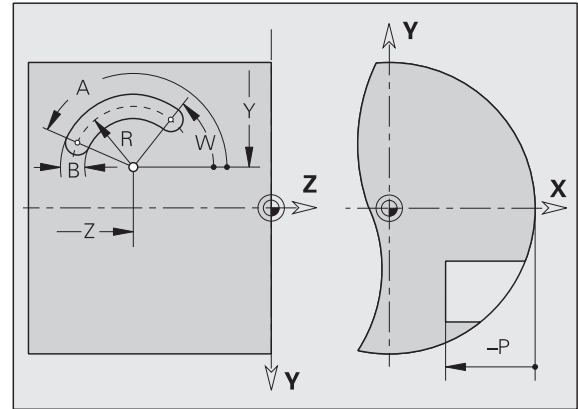
Circular slot G382/G383 Geo

G382/G383 defines a circular slot in the YZ plane.

- G382: Circular slot clockwise
- G383: Circular slot counterclockwise

Parameters

- Y Center of slot curvature
- Z Center of slot curvature
- X Reference diameter
 - No entry: "X" from section code
 - "X" from G381 overwrites "X" from section code
- R Radius; reference: center point path of the slot
- A Starting angle; reference: X axis (default: 0°)
- W End angle; reference: X axis (default: 0°)
- B Slot width
- P Pocket depth (default: "P" from G308)

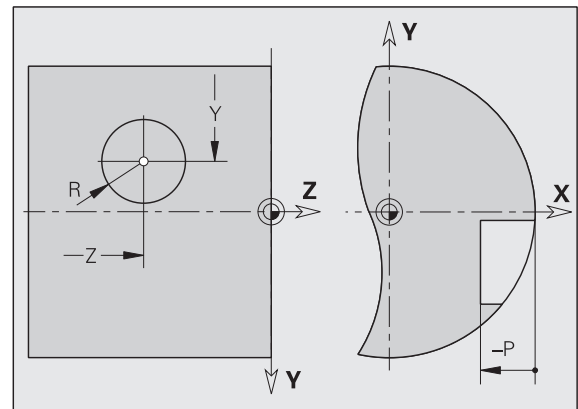


Full circle G384 Geo

G384 defines a full circle in the YZ plane.

Parameters

- Y Circle center
- Z Circle center
- X Reference diameter
 - No entry: "X" from section code
 - "X" from G381 overwrites "X" from section code
- R Circle radius
- P Pocket depth (default: "P" from G308)

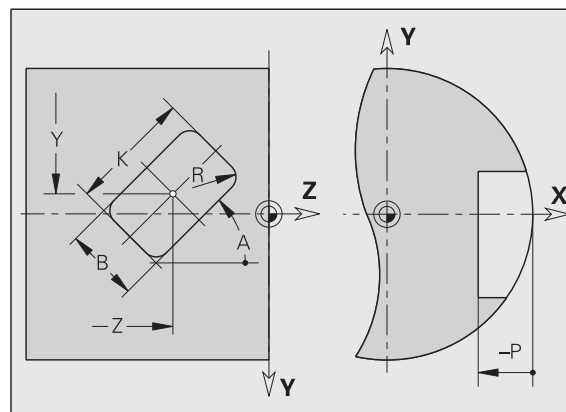


Rectangle G385 Geo

G385 defines a rectangle in the YZ plane.

Parameters

- Y Center of rectangle
- Z Center of rectangle
- X Reference diameter
 - No entry: "X" from section code
 - "X" from G381 overwrites "X" from section code
- A Angle of longitudinal axis to Z axis (default: 0°)
- K Length of rectangle
- B (Height) width of rectangle
- R Chamfer/rounding (default: 0)
 - $R > 0$: Radius of rounding arc
 - $R < 0$: Chamfer width
- P Pocket depth (default: "P" from G308)

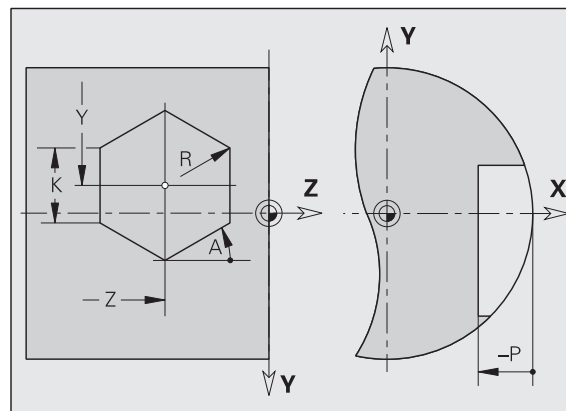


Eccentric polygon G387 Geo

G387 defines the contour of an eccentric polygon in the YZ plane.

Parameters

- Y Center point of polygon
- Z Center point of polygon
- X Reference diameter
 - No entry: "X" from section code
 - "X" from G381 overwrites "X" from section code
- Q Number of edges ($Q \geq 3$)
- A Angle to Z axis (default: 0°)
- K Edge length
 - $K > 0$: Edge length
 - $K < 0$: Inside diameter
- R Chamfer/rounding—default: 0
 - $R > 0$: Radius of rounding arc
 - $R < 0$: Chamfer width
- P Pocket depth (default: "P" from G308)



Linear pattern in YZ plane, G481-Geo

G481 defines a linear pattern in the YZ plane. G481 is effective for the figure defined in the following block (G380 to G385, G387).

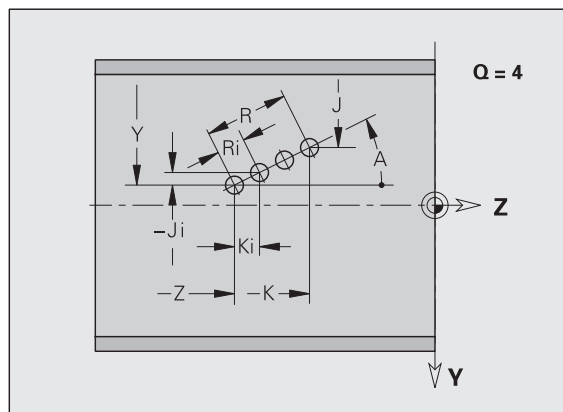
Parameters

Q	Number of figures
Y	Starting point of pattern
Z	Starting point of pattern
J	End point of pattern (Y direction)
K	End point of pattern (Z direction)
Ji	Distance between two figures (in Y direction)
Ki	Distance between two figures (in Z direction)
A	Angle of longitudinal axis of pattern (reference: positive Z axis)
R	Total length of pattern
Ri	Distance between two figures (pattern distance)



Programming notes

- Program the hole/figure in the following block without a center.
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.



Circular pattern in YZ plane, G482-Geo

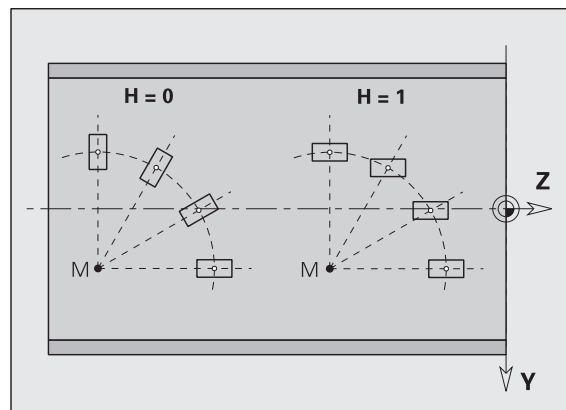
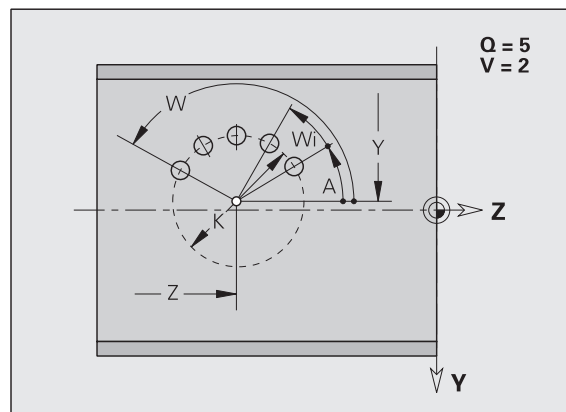
G482 defines a circular pattern in the YZ plane. G482 is effective for the figure defined in the following block (G380 to G385, G387).

Parameters

- Q Number of figures
- K Pattern diameter
- A Starting angle—position of the first figure; reference: Z axis (default: 0°)
- W End angle—position of the last figure; reference: Z axis (default: 360°)
- Wi Angle between two figures
- V Direction—orientation (default: 0)
 - V=0, without W: Figures are arranged on a full circle
 - V=0, with W: Figures are arranged on the longer circular arc
 - V=0, with Wi: The algebraic sign of Wi defines the direction (Wi<0: clockwise)
 - V=1, with W: Clockwise
 - V=1, with Wi: Clockwise (algebraic sign of Wi has no effect)
 - V=2, with W: Counterclockwise
 - V=2, with Wi: Counterclockwise (algebraic sign of Wi has no effect)
- Y Center of pattern
- Z Center of pattern
- H Position of the figures (default: 0)
 - H=0: Normal position; the figures are rotated about the circle center (rotation)
 - H=1: Original position; the position of the figures relative to the coordinate system remains unchanged (translation)



- Program the hole/figure in the following block without a center. **Exception: circular slot.**
- The milling cycle (MACHINING section) calls the hole/figure in the following block—not the pattern definition.



Single surface G386 Geo

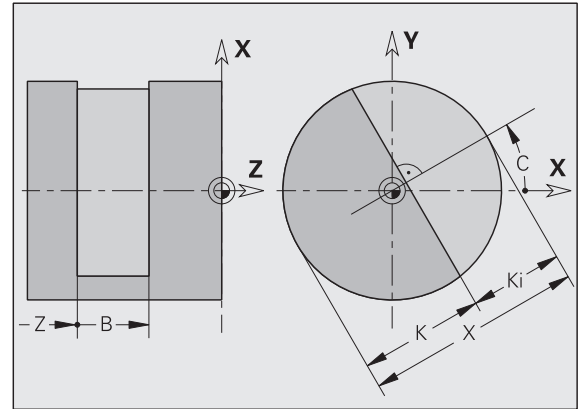
G386 defines a surface in the YZ plane.

Parameters

- Z Reference edge
- K Residual depth
- Ki Depth
- B Width (reference: reference edge Z)
 - $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction
- X Reference diameter
 - No entry: "X" from section code
 - "X" from G381 overwrites "X" from section code
- C Angular position of surface (default: "C" from section code)



The **reference diameter X** limits the surface to be machined.



Centric polygon G487 Geo

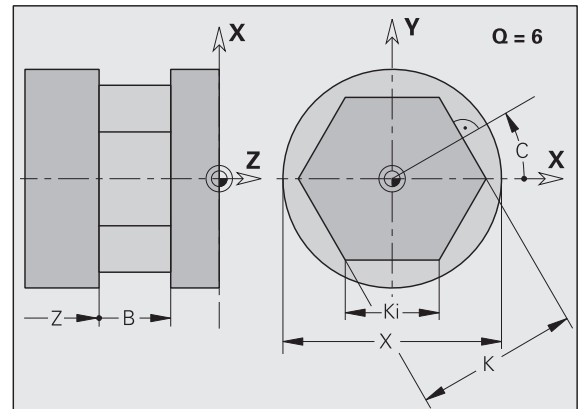
G487 defines polygonal surfaces in the YZ plane.

Parameters

- Z Reference edge
- K Inside diameter (width across flats)
- Ki Length of side
- B Width (reference: reference edge Z)
 - $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction
- X Reference diameter
 - No entry: "X" from section code
 - "X" from G381 overwrites "X" from section code
- C Angular position of surface (default: "C" from section code)
- Q Number of sides ($Q \geq 2$)



The **reference diameter X** limits the surface to be machined.



1.7 DIN PLUS: Working Planes

When programming drilling or milling operations with the Y axis, you need to define the working plane.

If no working plane is programmed, the CNC PILOT assumes a turning operation or a milling operation with the C axis (G18 XZ plane).

As of software version 625 952-05: At the end of a machining program (M30, M99) the working plane is reset to G18.

G17 XY plane (front or back)

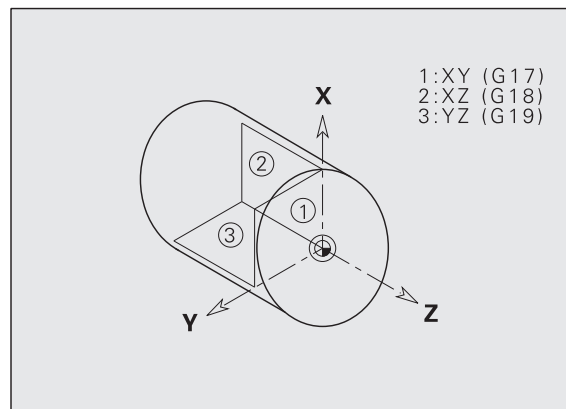
Milling cycles are executed in the XY plane, with the depth feed for milling and drilling cycles in the Z direction.

G18 XZ plane (turning)

In the XZ plane, "normal turning operations" as well as drilling and milling operations are executed with the C axis.

G19 YZ plane (side view/surface)

Milling cycles are executed in the YZ plane, with the depth feed for milling and drilling cycles in the X direction.



Tilting the working plane G16

G16 executes the following transformations and rotations:

- Shifts the coordinate system to the position I, K
- Rotates the coordinate system by the angle B; reference point: I, K
- Shifts, if programmed, the coordinate system by U and W in the rotated coordinate system

Parameters

- B Plane angle; reference: positive Z axis
- I Plane reference in X direction (radius)
- K Plane reference in Z direction
- U Shift in X direction
- W Shift in Z direction
- Q Enable/disable tilting the working plane
- 0: Disable tilted working plane function
 - 1: Tilt working plane
 - 2: Restore previous G16 plane

G16 Q0 resets the working plane. The zero point and coordinate system defined before G16 are then in effect again.

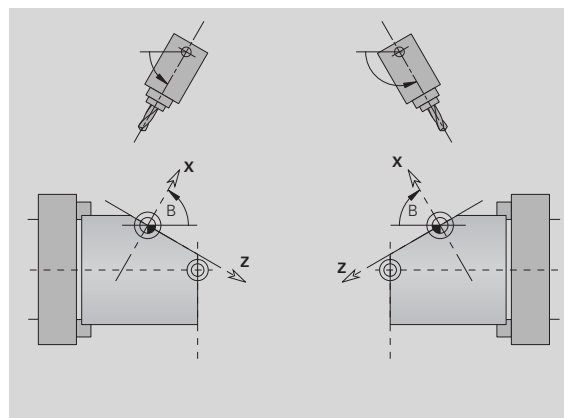
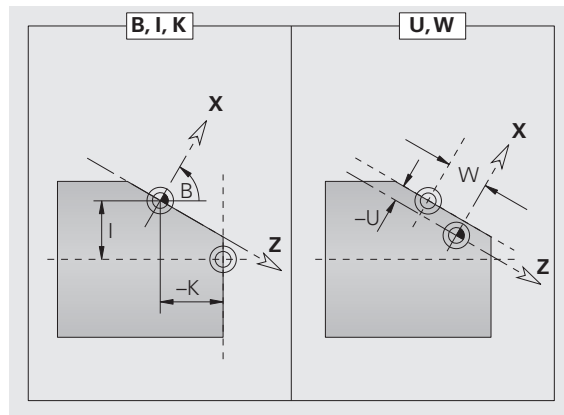
G16 Q2 restores the previous G16 plane.

The positive Z axis is the reference axis for the "plane angle B." This also applies to a mirrored coordinate system.



Please note:

- X is the infeed axis in a tilted coordinate system. X coordinates are entered as diameter coordinates.
- Mirroring the coordinate system has no effect on the reference axis of the tilt angle ("B axis angle" of G714).
- Other zero point shifts are not permitted while G16 is active.



Example: "G16"

```

. . .
BEARBEITUNG [MACHINING]
...
N.. G19
N.. G15 B130
N.. G16 B130 I59 K0 Q1
N.. G1 X.. Z.. Y..
N.. G16 Q0
. . .

```

1.8 DIN PLUS (Y Axis): Positioning Commands

Rapid traverse G0

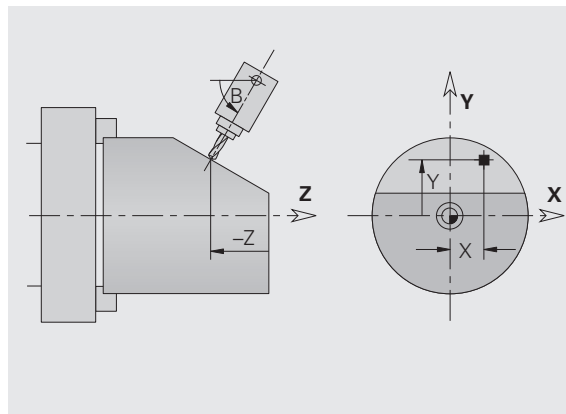
G0 moves the tool at rapid traverse along the shortest path to the target point X, Y, Z and tilts the B axis.

Parameters

- X Diameter—target point
Z Length—target point
Y Length—target point
B Angle of the B axis



Programming X, Y, Z, B: Absolute, incremental or modal



Approach tool change point G14

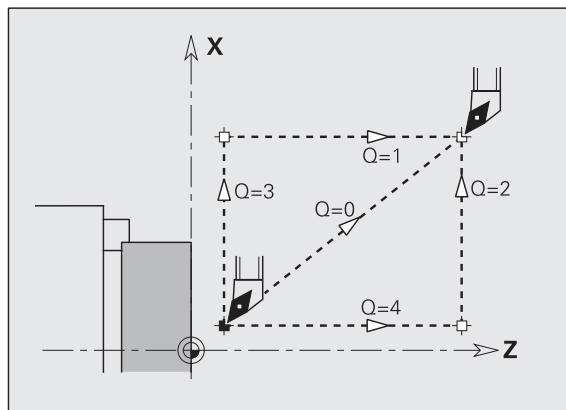
G14 moves the slide at rapid traverse to the tool change position. In setup mode, define permanent coordinates for the tool change position.

Parameters

- Q Sequence (default: 0)
- 0: Move simultaneously in X and Z axes (diagonal path)
 - 1: First X, then Z direction
 - 2: First Z, then X direction
 - 3: Only X direction, Z remains unchanged
 - 4: Only Z direction, X remains unchanged
 - 5: Y direction only
 - 6: Move simultaneously in X, Y and Z axes (diagonal path)



If Q=0 to 4, the Y axis does not move.



Rapid traverse to machine coordinates G701

G701 moves the tool at rapid traverse along the shortest path to the target point X, Y, Z and tilts the B axis.

Parameters

X	End point (diameter)
Y	End point
Z	End point
B	Angle of the B axis



“X, Y, Z” refer to the **machine zero point** and the **slide reference point**.

1.9 DIN PLUS: Magazine Tools

Insert magazine tool G714



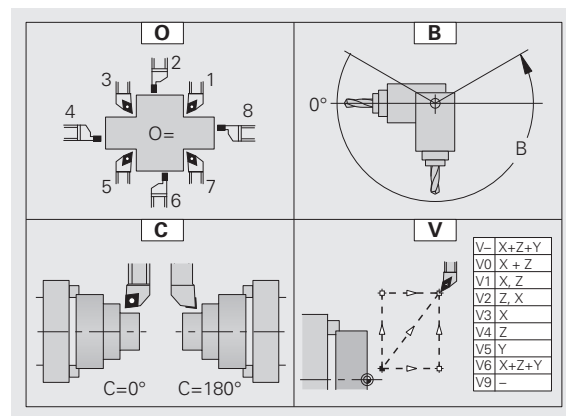
Cycle G714 is interfaced to the control and the machine by the machine tool builder. The following description of the parameters and the operating sequence may therefore deviate from the functionality provided on your machine. The machine manual provides more detailed information.

G714 contains the following functions:

- Move to the tool change position.
- Return the active tool to the magazine.
- Take the programmed tool from the magazine.
- Tilt the B axis to the programmed angle.
- Rotate the tool to the "position angle" ("normal" or "upside down" attitude).
- Calculate the tool data on the basis of the "orientation O," the B axis position and the position angle.
- If programmed, activate the (additive) "compensation D."
- Set the shoe brake, as programmed.

Parameters

- ID ID number of the magazine tool.
- Press the "Continue" soft key to display the plate list.
Highlight the required tool and confirm your selection.
- O Orientation of turning tools. Position of the tool's cutting edge (see figure).
- Tool positions 1, 3, 5, 7: For roughing, finishing and button tools (neutral tools are recognized by the tool angle).
 - Tool positions 2, 4, 6, 8: For recessing and threading cycles (a "right-hand" or "left-hand" tool is defined in the tool data).
- B B axis angle. Angle to which the B axis is tilted.
- C Offset position angle of turning tools.
- 0°: Tool attitude "normal"
 - 180°: Tool attitude "upside down"
- D Additive compensation (1 to 16). Activates the additive compensation. The additive compensation is deactivated with the next tool change (see G149).
- H Shoe brake
- 0: The brake is locked depending on the tool parameter (if "not driven" the brake is locked; if "driven" the brake is not locked)
 - 1: The brake is locked
 - 2: The brake is not locked



Parameters

- V Approach the tool change position (default: 6)
- No entry: Move simultaneously in X, Y and Z directions.
 - 0: Move simultaneously in X and Z directions
 - 1: First X, then Z direction
 - 2: First Z, then X direction
 - 3: X direction only
 - 4: Z direction only
 - 5: Y direction only
 - 6: Move simultaneously in X, Y and Z directions
 - 9: Do **not** approach the tool change position

As of software version 625 952-02:

Parameters

- Q Additional functions
- You can use this parameter to transfer a value to the tool change. The function of this value is defined by the machine tool builder.
- X Diameter
- X diameter to which the tool moves at the end of the tool change, if required.
- Z Length
- Z position to which the tool moves at the end of the tool change, if required.
- Y Length
- Y position to which the tool moves at the end of the tool change, if required.

Returning the tool to the magazine: If you program G714 without an "ID number," the CNC PILOT will return the active tool to the magazine without inserting a new tool.

Changing the tool position: If the call refers to the active tool, the B axis is tilted and/or the position angle is changed. In the parameter "V," you can define whether this function is to be executed at the current position or at the tool change position.

Offset position angle: With the "offset position angle" you can position turning tools in the "normal" attitude or "upside down." When positioning the tool, the CNC PILOT also takes the basic setting saved in the tool database into account (position angle = position angle from the tool data + offset position angle).

Tool orientation: The CNC PILOT takes the orientation of the cutting edge into account when calculating the position of the tool tip. The control distinguishes between roughing, finishing and button tools as well as recessing and threading tools (see figure).

G16 active: If a tilted plane (G16) is active, it is deactivated while the G714 call is executed. The tilted plane is then effective again after G714.



HEIDENHAIN recommends using G714 also for changing the tilt angle or tool position (offset position angle).

Example: G714

. . .	
FERTIGTEIL [FINISHED PART]	
. . .	
SURFACE_Y X118 C0 B130 I59 K0	Describe the tilted working plane
. . .	
BEARBEITUNG [MACHINING]	
. . .	
N . . G714 ID"B_522-32-10" 00 B130	Insert the magazine tool; tilt the B axis
N . . G19	Activate the YZ plane
. . .	
N . . G840 NS ..	Milling operation in tilted plane
. . .	
N . . G18	Activate the XZ plane
N . . G714 ID"B_112-93-80" 01 B90 C0	Insert the magazine tool; tilt the B axis; set the offset position angle for the tool
. . .	
N . . G810 NS ..	Turning
. . .	
ENDE [END]	

Define tool position G712



Cycle G712 is interfaced to the control and the machine by the machine tool builder. The following description of the parameters and the operating sequence may therefore deviate from the functionality provided on your machine. The machine manual provides more detailed information.

If the B axis was positioned using single commands, you need to inform the control of the tool position by programming G712.

G712 contains the following declarations:

- Angle of the B axis
- Offset position angle
- Tool orientation

Parameters

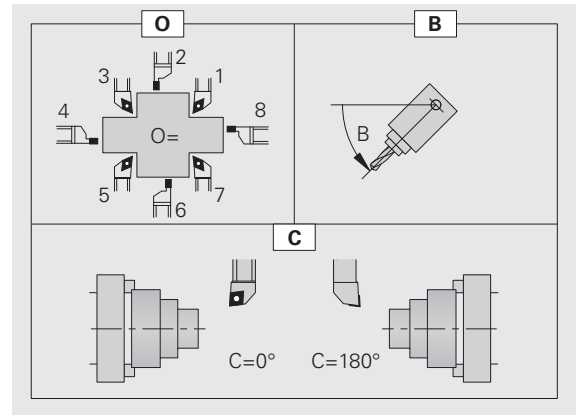
- B B axis angle. Angle at which the B axis is located.
- C Offset position angle of turning tools.
- 0°: Tool attitude "normal"
 - 180°: Tool attitude "upside down"
- O Orientation of turning tools. Position of the tool's cutting edge (see figure).
- Tool positions 1, 3, 5, 7: For roughing, finishing and button tools (neutral tools are recognized by the tool angle).
 - Tool positions 2, 4, 6, 8: For recessing and threading cycles (a "right-hand" or "left-hand" tool is defined in the tool data).

Offset position angle: The "offset position angle" rotates turning tools to the "normal" or "upside down" attitude. When positioning the tool, the CNC PILOT also takes the basic setting saved in the tool database into account (position angle = position angle from the tool data + offset position angle).

Tool orientation: The CNC PILOT takes the orientation of the cutting edge into account when calculating the position of the tool tip. The control distinguishes between roughing, finishing and button tools as well as recessing and threading tools (see figure).



G712 defines the position of the tool. The tool is **not** moved.



Preselect tool G600

As of software version 625 952-04:

If your lathe is equipped with a tool magazine, you can use G600 for special functions. G600 transfers the current magazine pocket number of the tool to the PLC.

Parameters

ID ID number of the magazine tool.



The G600 function is provided by the machine tool builder.
The machine manual provides more detailed information.

1.10 DIN PLUS: Linear and Circular Paths

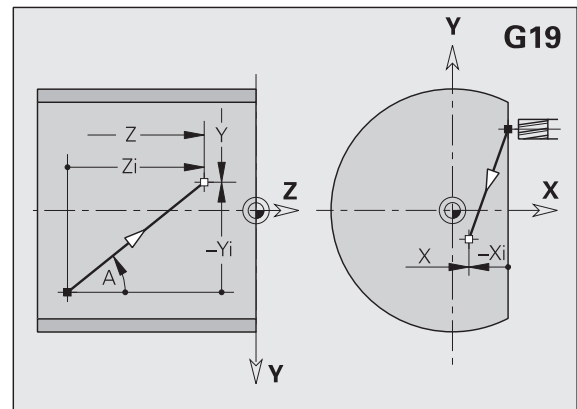
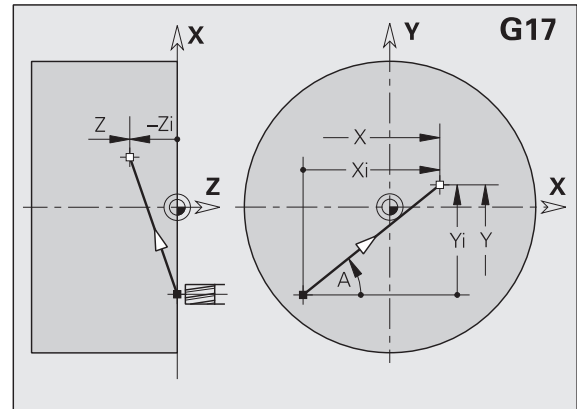
Milling: Linear movement G1

G1 moves the tool on a linear path at the feed rate to the "end point."
The execution of G1 varies depending on the **working plane**:

- G17 Interpolation in the XY plane
 - Infeed in Z direction
 - Angle A—reference: positive X axis
- G18 Interpolation in the XZ plane
 - Infeed in Y direction
 - Angle A—reference: negative Z axis
- G19 Interpolation in the YZ plane
 - Infeed in X direction
 - Angle A—reference: positive Z axis

Parameters

- X End point (diameter)
Y End point
Z End point
A Angle (reference: depends on the working plane)
Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
- Q=0: Near point of intersection
 - Q=1: Far point of intersection
- B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
- No entry: Tangential transition
 - B=0: No tangential transition
 - B>0: Rounding radius
 - B<0: Chamfer width
- E Special feed factor for the chamfer/rounding (default: 1)
Special feed rate = active feed rate * E ($0 < E \leq 1$)



Programming X, Y, Z: Absolute, incremental or modal or "?"

Milling: Circular movement G2, G3—incremental center coordinates

G2/G3 moves the tool in a circular arc at the feed rate to the “end point.”

The execution of G2/G3 varies depending on the **working plane**:

- G17 Interpolation in the XY plane
 - Infeed in Z direction
 - Center definition: with I, J
- G18 Interpolation in the XZ plane
 - Infeed in Y direction
 - Center definition: with I, K
- G19 Interpolation in the YZ plane
 - Infeed in X direction
 - Center definition: with J, K

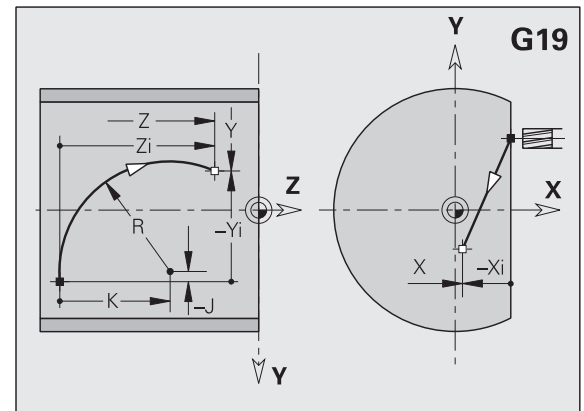
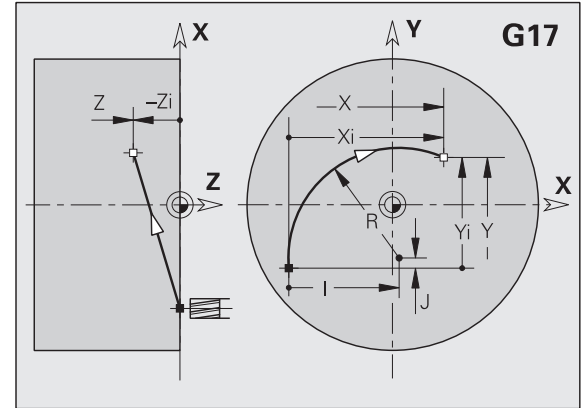
Parameters

- X End point (diameter)
- Y End point
- Z End point
- I Incremental center point (radius)
- J Incremental center point
- K Incremental center point
- R Radius
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
- Q=0: Near point of intersection
 - Q=1: Far point of intersection
- B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
- No entry: Tangential transition
 - B=0: No tangential transition
 - B>0: Rounding radius
 - B<0: Chamfer width
- E Special feed factor for the chamfer/rounding (default: 1)
- Special feed rate = active feed rate * E (0 < E ≤ 1)

If you do not program the center, the CNC PILOT automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.



Programming X, Y, Z: Absolute, incremental or modal or “?”



Milling: Circular path G12, G13—absolute center coordinates

G12/G13 moves the tool in a circular arc at the feed rate to the “end point.”

The execution of G12/G13 varies depending on the **working plane**:

- G17 Interpolation in the XY plane
 - Infeed in Z direction
 - Center definition: with I, J
- G18 Interpolation in the XZ plane
 - Infeed in Y direction
 - Center definition: with I, K
- G19 Interpolation in the YZ plane
 - Infeed in X direction
 - Center definition: with J, K

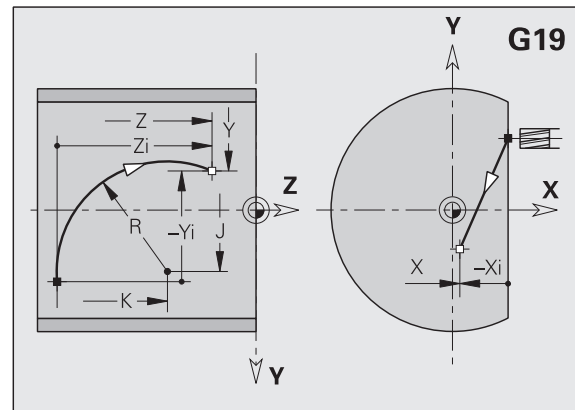
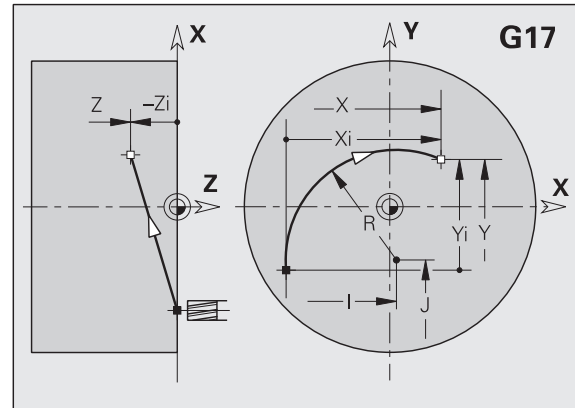
Parameters

- X End point (diameter)
- Y End point
- Z End point
- I Absolute center point (radius)
- J Absolute center point
- K Absolute center point
- R Radius
- Q Point of intersection. End point if the line segment intersects a circular arc (default: 0):
- Q=0: Near point of intersection
 - Q=1: Far point of intersection
- B Chamfer/rounding. Defines the transition to the next contour element. When entering a chamfer/rounding, program the theoretical end point.
- No entry: Tangential transition
 - B=0: No tangential transition
 - B>0: Rounding radius
 - B<0: Chamfer width
- E Special feed factor for the chamfer/rounding (default: 1)
- Special feed rate = active feed rate * E (0 < E ≤ 1)

If you do not program the center, the CNC PILOT automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.



Programming X, Y, Z: Absolute, incremental or modal or “?”



1.11 DIN PLUS (Y Axis): Milling Cycles

Area milling—roughing G841

G841 roughs surfaces defined with G376 Geo (XY plane) or with G386 Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

Parameters

- NS Block number—reference to the contour description
- P (Maximum) milling depth (infeed in the working plane)
- I Oversize in X direction
- K Oversize in Z direction
- U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).
- Overlap = $U \times \text{milling diameter}$
- V Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5).
- Overrun = $V \times \text{milling diameter}$
- F Feed rate for infeed (default: active feed rate)
- J Retraction plane (default: back to starting position)
- XY plane: Retraction position in Z direction
 - YZ plane: Retraction position in X direction (diameter)

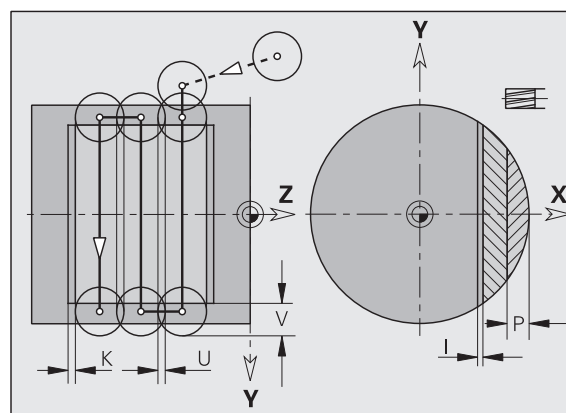
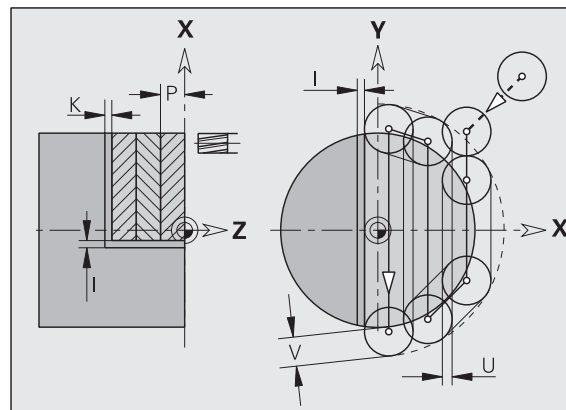


Oversizes are taken into account:

- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane

Cycle run

- 1 Starting position (X, Y, Z, C) is the position before the cycle begins.
- 2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths).
- 3 Move to the safety clearance and plunge to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete area is milled.
- 7 Return to "retraction plane J."



Area milling—finishing G842

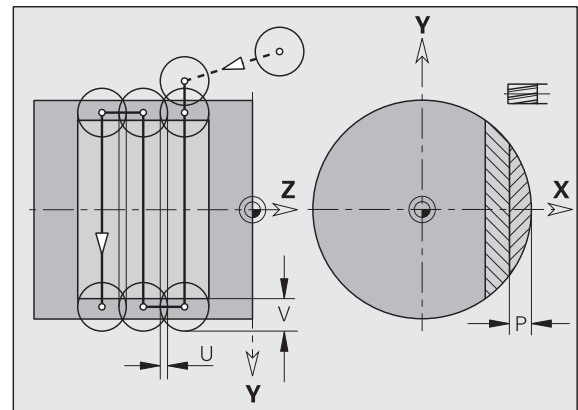
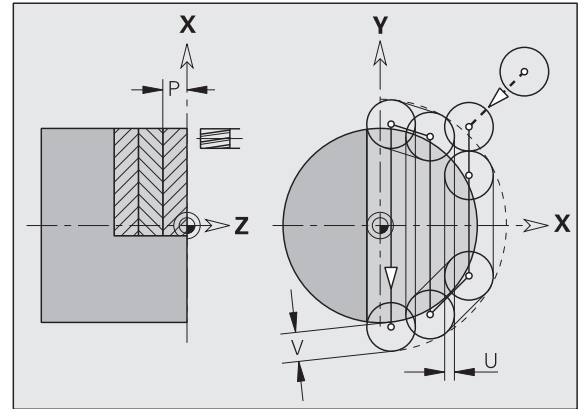
G842 finishes surfaces defined with G376 Geo (XY plane) or G386 Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

Parameters

- NS Block number—reference to the contour description
- H Cutting direction for side finishing (default: 0)
- H=0: Up-cut milling
 - H=1: Climb milling
- P (Maximum) milling depth (infeed in the working plane)
- U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).
- Overlap = $U \times \text{milling diameter}$
- V Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5).
- Overrun = $V \times \text{milling diameter}$
- F Feed rate for infeed (default: active feed rate)
- J Retraction plane (default: back to starting position)
- XY plane: Retraction position in Z direction
 - YZ plane: Retraction position in X direction (diameter)

Cycle run

- 1 Starting position (X, Y, Z, C) is the position before the cycle begins.
- 2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths).
- 3 Move to the safety clearance and plunge to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete area is milled.
- 7 Return to "retraction plane J."



Centric polygon milling—roughing G843

G843 roughs centric polygons defined with G477 Geo (XY plane) or G487 Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

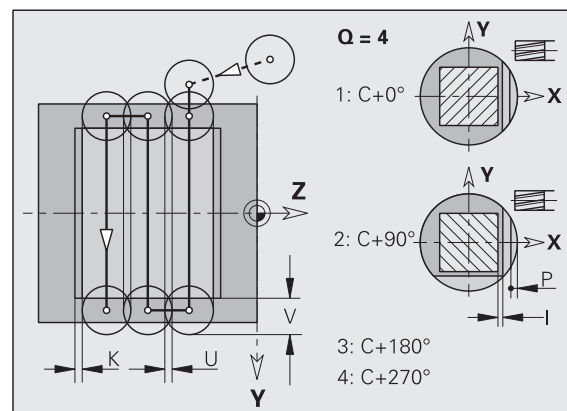
Parameters

- NS Block number—reference to the contour description
- P (Maximum) milling depth (infeed in the working plane)
- I Oversize in X direction
- K Oversize in Z direction
- U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).
- Overlap = $U \times \text{milling diameter}$
- V Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5).
- Overrun = $V \times \text{milling diameter}$
- F Feed rate for infeed (default: active feed rate)
- J Return plane (default: back to starting position)
- XY plane: Retraction position in Z direction
 - YZ plane: Retraction position in X direction (diameter)



Oversizes are taken into account:

- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane



Cycle run

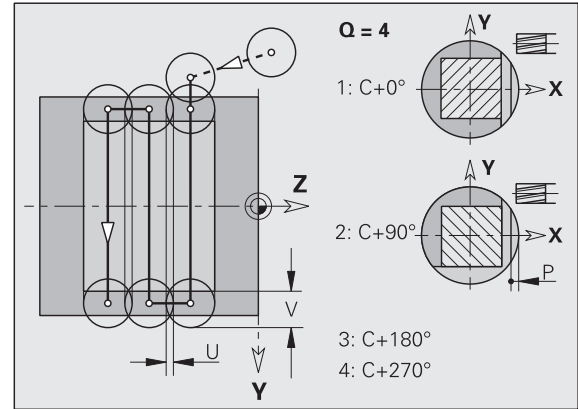
- 1 Starting position (X, Y, Z, C) is the position before the cycle begins.
- 2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the spindle positions.
- 3 Spindle turns to the first position. The tool moves to the safety clearance and plunges to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete area is milled.
- 7 The tool returns to "retraction plane J." The spindle turns to the next position. The tool moves to the safety clearance and plunges to the first milling depth.
- 8 Repeat steps 4 to 7 until all polygonal surfaces are milled.
- 9 Return to "retraction plane J."

Centric polygon milling—finishing G844

G844 finishes centric polygons defined with G477 Geo (XY plane) or with G487 Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

Parameters

- NS Block number—reference to the contour description
- H Cutting direction for side finishing (default: 0)
- H=0: Up-cut milling
 - H=1: Climb milling
- P (Maximum) milling depth (infeed in the working plane)
- U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).
- Overlap = $U \times \text{milling diameter}$
- V Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5).
- Overrun = $V \times \text{milling diameter}$
- F Feed rate for infeed (default: active feed rate)
- J Retraction plane (default: back to starting position)
- XY plane: Retraction position in Z direction
 - YZ plane: Retraction position in X direction (diameter)



Cycle run

- 1 Starting position (X, Y, Z, C) is the position before the cycle begins.
- 2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the spindle positions.
- 3 Spindle turns to the first position. The tool moves to the safety clearance and plunges to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete area is milled.
- 7 The tool returns to "retraction plane J." The spindle turns to the next position. The tool moves to the safety clearance and plunges to the first milling depth.
- 8 Repeat steps 4 to 7 until all polygonal surfaces are milled.
- 9 Return to "retraction plane J."

Pocket milling - roughing G845 (Y axis)

G845 roughs closed contours that are defined in the XY or YZ plane in the program sections:

- STIRN_Y [FRONT_Y]
- RUECKSEITE_Y [REAR_SIDE_Y]
- MANTEL_Y [SURFACE_Y]

Choose one of the following **plunge strategies**, depending on the milling cutter you are using:

- Plunge vertically
- Plunge at a pre-drilled position
- Plunge in a reciprocating or helical motion

When “plunging at a pre-drilled position,” you have the following alternatives:

- **Calculate positions, drill, mill.** The machining process is performed in the following steps:
 - Insert drill.
 - Calculate hole positions with “G845 A1 ...”
 - Drill holes with “G71 NF ...”
 - Call cycle “G845 A0 ...” The cycle positions the tool above the hole; the tool plunges and mills the pocket.
- **Drill, mill.** The machining process is performed in the following steps:
 - Drill a hole inside the pocket with “G71 ...”
 - Position the milling cutter above the hole and call “G845 A0 ...” The tool plunges and mills the section.

If the pocket consists of multiple sections, G845 takes all the sections of the pocket into account for drilling and milling. Call “G845 A0 ...” separately for each section when calculating the hole positions without “G845 A1 ...”



G845 takes the following oversizes into account:

- G57: Oversize in X, Z direction
- G58: Equidistant oversize in the milling plane

Program oversizes for calculating the hole positions **and** for milling.

G845 (Y axis) – Calculating hole positions

“G845 A1 ..” calculates the hole positions and stores them at the reference specified in “NF.” The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling “G845 A1 ..”. Program only the parameters given in the following list.

See also:

- G845—Fundamentals: Page 56
- G845—Milling: Page 58

Parameters—Calculating hole positions

NS	Block number—reference to the contour description
I	Oversize in X direction
K	Oversize in Z direction
Q	Machining direction (default: 0) <ul style="list-style-type: none"> ■ Q=0: From the inside toward the outside ■ Q=1: From the outside toward the inside
A	Sequence for “Calculate hole positions”: A=1
NF	Position mark—reference at which the cycle stores the hole positions [1 to 127].
WB	Plunge length—diameter of the milling cutter



- G845 overwrites any hole positions that may still be stored at the reference “NF.”
- The parameter “WB” is used both for calculating the hole positions and for milling. When calculating the hole positions, “WB” describes the diameter of the milling cutter.

G845 (Y axis) – Milling

You can change the **cutting direction** with the “cutting direction H,” the “machining direction Q” and the direction of tool rotation (see table G845 in the User’s Manual). Program only the parameters given in the following list.

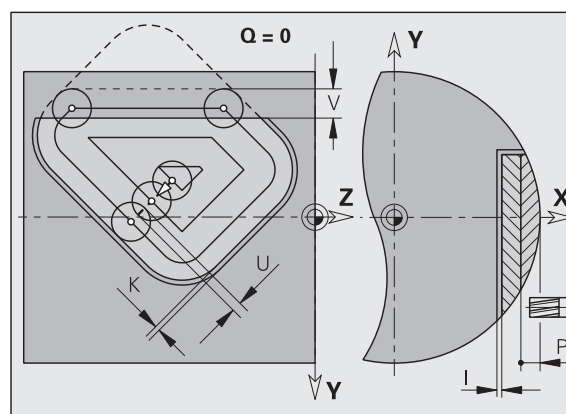
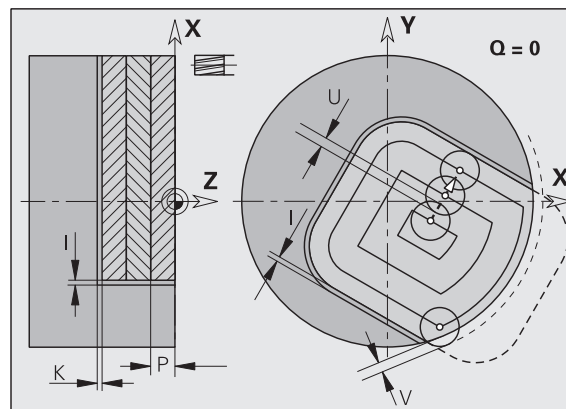
See also:

- G845—Fundamentals: Page 56
- G845—Calculating hole positions: Page 57

Parameters—milling

- NS Block number—reference to the contour description
- P (Maximum) milling depth (infeed in the working plane)
- I Oversize in X direction
- K Oversize in Z direction
- U (Minimum) overlap factor (default: 0.5)
Overlap = $U \times \text{milling diameter}$
- V Overrun factor. Defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5):
- 0: The defined contour is milled completely
 - $0 < V \leq 1$: Overrun = $V \times \text{milling diameter}$
- H Cutting direction (default: 0)
- H=0: Up-cut milling
 - H=1: Climb milling
- F Feed rate for infeed (default: active feed rate)
- Starting with software version 625 952-05: F is used as machining feed rate for reciprocating or helical plunging.
- E Reduced feed rate for circular elements (default: current feed rate)
- J Retraction plane (default: back to starting position)
- XY plane: Retraction position in Z direction
 - YZ plane: Retraction position in X direction (diameter)
- Q Machining direction (default: 0)
- Q=0: From the inside toward the outside
 - Q=1: From the outside toward the inside
- A Sequence for “Milling”: A=0 (default=0)
- NF Position mark—reference from which the cycle reads the hole positions [1 to 127].
- O Approach behavior (default: 0)

Plunge vertically O=0: The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the pocket.



Parameters—milling**Plunge at pre-drilled position O=1:**

- If "NF" is programmed: The cycle positions the milling cutter above the first pre-drilled hole; the tool plunges and mills the first area. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next area, etc.
- If "NF" is not programmed: The tool plunges at the current position and mills the area. If applicable, position the tool to the next pre-drilled hole and mill the next area, etc.

Plunge in a helical motion O=2, 3: The tool plunges at the angle "W" and mills full circles with the diameter "WB." As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=2—manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=3—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends on the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas.

Plunge in a reciprocating linear motion O=4, 5: The tool plunges at the angle "W" and mills a linear path of the length "WB." You can define the position angle in "WE." The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling.

- O=4—manually: The cycle plunges at the current position and machines the area that can be reached from this position.
- O=5—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends on the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas. The plunging position is determined from the type of figure and from "Q:"
 - Q0 (from the inside toward the outside):
 - Linear slot, rectangle, polygon: Reference point of the figure
 - Circle: Circle center
 - Circular slot, "free" contour: Starting point of the innermost milling path
 - Q1 (from the outside toward the inside):
 - Linear slot: Starting point of the slot
 - Circular slot, circle: Not machined
 - Rectangle, polygon: Starting point of the first linear element
 - "Free" contour: Starting point of the first linear element (at least one linear element must exist)

Parameters—milling

Plunge in a reciprocating circular motion O=6, 7: The tool plunges at the plunging angle "W" and mills a circular arc of 90°. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth "P," the cycle switches to face milling. "WE" defines the arc center, "WB" the arc radius.

- O=6—manually: The tool position corresponds to the center of the circular arc. The tool moves to the arc starting point and plunges.
- O=7—automatically (only permitted for circular slots and circles): The cycle calculates the plunging position on the basis of "Q:"
 - Q0 (from the inside toward the outside):
 - Circular slot: The circular arc lies on the curvature radius of the slot
 - Circle: Not permitted
 - Q1 (from the outside toward the inside): Circular slot, circle: The circular arc lies on the outermost milling path

W Plunging angle in infeed direction

WE Position angle of the milling path/circular arc. Reference axis:

- Face or back: Positive XK axis
- Lateral surface: Positive Z axis

Default position angle, depending on "O:"

- O=4: WE= 0°
- O=5 and
 - Linear slot, rectangle, polygon: WE= position angle of the figure
 - Circular slot, circle: WE=0°
 - "Free" contour and Q0 (from the inside toward the outside): WE=0°
 - "Free" contour and Q1 (from the outside toward the inside): Position angle of the starting element

WB Plunge length/plunge diameter (default: 1.5 * milling diameter)

For the cutting direction, machining direction and direction of tool rotation, please refer to table G845 in the User's Manual.



For the machining direction Q=1 (from the outside toward the inside), please note:

- The contour must start with a linear element.
- If the starting element is < WB, WB is reduced to the length of the starting element.
- The length of the starting element must not be less than 1.5 times the diameter of the milling cutter.

Cycle run

- 1 Starting position (X, Y, Z, C) is the position before the cycle begins.
- 2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the plunging paths for reciprocating or helical plunges.
- 3 Move to the safety clearance and plunge to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete area is milled.
- 7 Return to "retraction plane J."

Pocket milling—finishing G846 (Y axis)

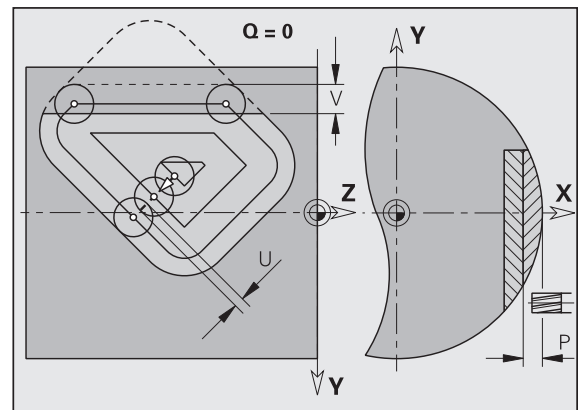
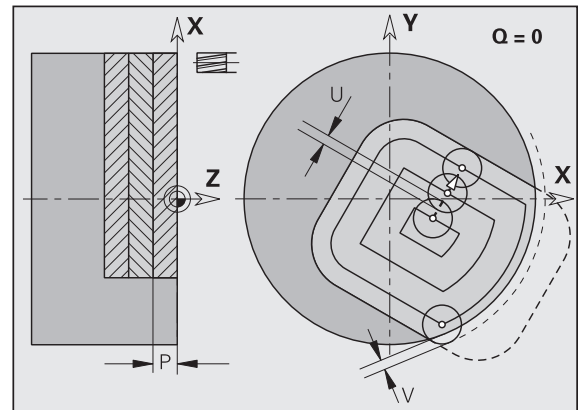
G846 finishes closed contours defined in the XY or YZ plane in the program sections:

- FACE_Y
- REAR_Y
- LATERAL_Y

You can change the **cutting direction** with the "cutting direction H," the "machining direction Q" and the direction of tool rotation.

Parameters—Milling

- NS Block number—reference to the contour description
- P (Maximum) milling depth (infeed in the working plane)
- R Radius of approaching/departing arc (default: 0)
- R=0: The contour element is approached directly; Feed to the starting point above the milling plane, then vertical plunge.
 - R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
- U (Minimum) overlap factor. Defines the overlap of milling paths (default: 0.5).
- Overlap = $U \times \text{milling diameter}$
- V Overrun factor if the milling contour lies partly outside the turning contour:
- 0: The defined contour is milled completely
 - $0 < V \leq 1$: Overrun = $V \times \text{milling diameter}$
- H Cutting direction (default: 0)
- H=0: Up-cut milling
 - H=1: Climb milling
- F Feed rate for infeed (default: active feed rate)



Parameters—Milling

- E Reduced feed rate for circular elements (default: current feed rate)
- J Retraction plane (default: back to starting position)
 - XY plane: Retraction position in Z direction
 - YZ plane: Retraction position in X direction (diameter)
- Q Machining direction (default: 0)
 - Q=0: From the inside toward the outside
 - Q=1: From the outside toward the inside
- O Plunging behavior (default: 0)
 - O=0—plunge vertically: The cycle moves the tool to the starting point; the tool plunges and finishes the pocket.
 - Q=1—Approaching arc with depth feed: When machining the upper milling planes, the tool advances to the milling plane and then approaches on an arc. When machining the bottom milling plane, the tool plunges to the milling depth while moving on the approaching arc (three-dimensional approaching arc). You can use this approach behavior only in conjunction with an approaching arc “R” and when machining from the outside toward the inside (Q=1).

For the cutting direction, machining direction and direction of tool rotation, please refer to table G846 in the User's Manual.

Cycle run

- 1 Starting position (X, Y, Z, C) is the position before the cycle begins.
- 2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths).
- 3 Move to the safety clearance and plunge to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete area is milled.
- 7 Return to “retraction plane J.”

Engrave in XY plane G803

G803 engraves character strings aligned linearly in the YZ plane. The text to be engraved is entered in the "ID" box as a character string.

Parameters

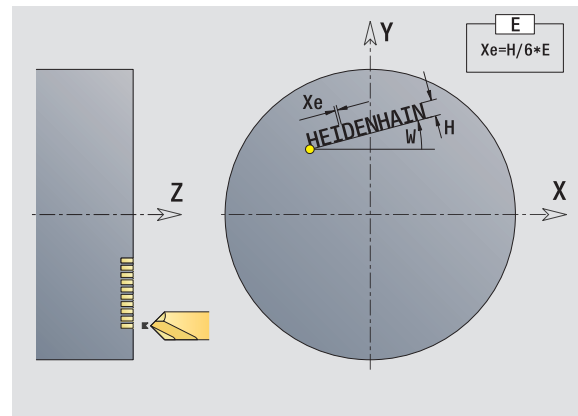
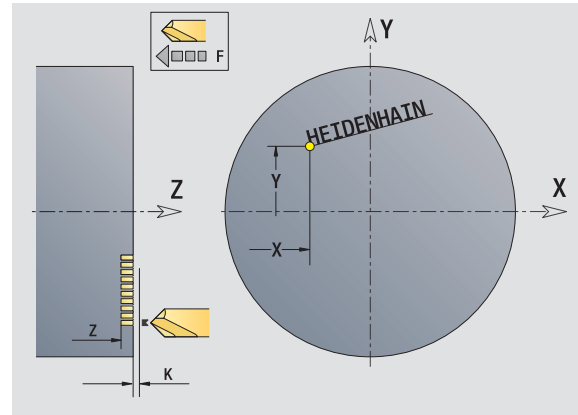
ID	Text. Text to be engraved ()
NS	Character number. ASCII code of the character to be engraved
X	Starting diameter (diameter dimension)
Y	Starting point
Z	Milling floor. Z position, infeed depth during milling.
K	Retraction plane. Z position retracted to for positioning.
H	Font height. Height of the characters in [mm]
W	Position angle of the character string with linear alignment. Example: 0° = Vertical characters: the characters are aligned in sequence in positive X direction
E	Distance factor. The distance between the characters is calculated according to the following formula: $H / 6 * E$
F	Available as of software version 625 952-05. Approach feed factor (feed rate = current feed rate * F)

Diacritics and special characters that you cannot enter in the DIN editor can be defined, character by character, in NS. If text is defined in "ID" and a character is defined in "NS," the text is engraved before the character.

G803 starts engraving from the starting position or from the current position, if no starting position is defined.

Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Character table: see the User's Manual.



Engrave in YZ plane G804

G804 engraves character strings aligned linearly in the YZ plane. The text to be engraved is entered in the "ID" box as a character string.

Parameters

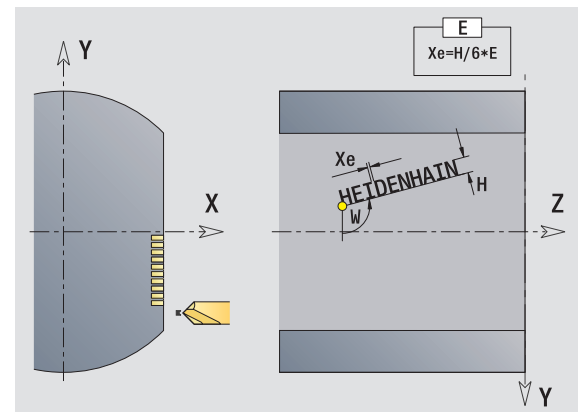
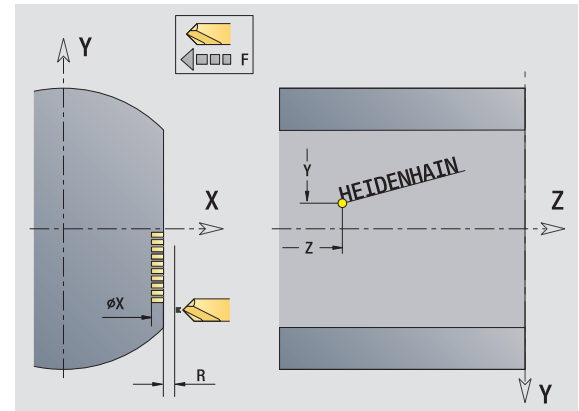
- ID Text. Text to be engraved ()
- NS Character number. ASCII code of the character to be engraved
- Z Starting point
- Y Starting point
- X Milling floor (diameter). X position, infeed depth during milling.
- I Retraction diameter. X position retracted to for positioning.
- H Font height. Height of the characters in [mm]
- W Position angle of the character string. Examples:
- 0°: from -Y to +Y
 - 90°: from -Z to +Z (see figure)
- E Distance factor. The distance between the characters is calculated according to the following formula: $H / 6 * E$
- F As of software version 625 952-05:
Approach feed factor (feed rate = current feed rate * F)

Diacritics and special characters that you cannot enter in the DIN editor can be defined, character by character, in "NS." If text is defined in "ID" and a character is defined in "NS," the text is engraved before the character.

G804 starts engraving from the starting position or from the current position, if no starting position is defined.

Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Character table: see User's Manual.



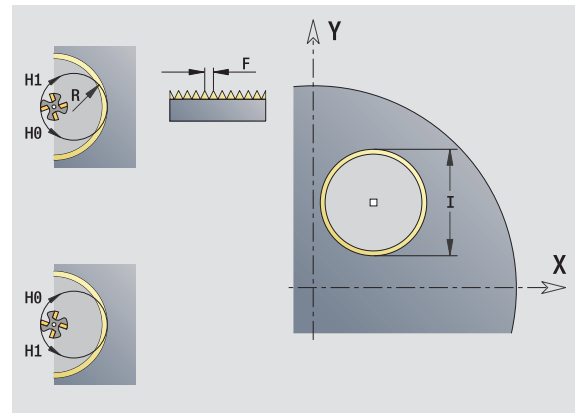
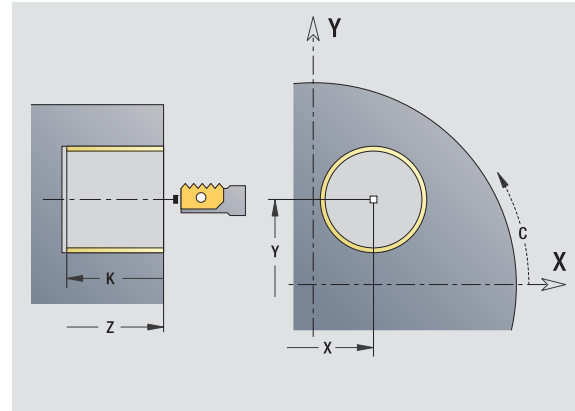
Thread milling in XY plane G800

Starting with software version 625 952-05: G800 mills a thread in existing holes.

The cycle positions the tool on the end point of the thread within the hole. Then the tool approaches on "approaching radius R" and mills the thread. During this, the tool advances by the thread pitch F. Following that, the cycle retracts the tool and returns it to the starting point. With parameter V, you can program whether the thread is to be milled in one revolution or, with single-point tools, in several revolutions.

Parameters

- X Starting point
- Y Starting point
- C Starting point
- Z Milling top edge
- I Thread diameter
- K Thread depth
- R Approach radius
- F Thread pitch
- J Direction of thread—(default: 0)
 - 0: Right-hand thread
 - 1: Left-hand thread
- H Cutting direction (default: 0)
 - 0: Up-cut milling
 - 1: Climb milling
- V One rotation / several rotations
 - 0: The thread is milled in a 360-degree rotation
 - 1: The thread is milled in several rotations (single-point tool)
- O Spindle brake (O is evaluated if the brake is entered in machine parameter 1019, ..)—default: 0
 - 0: Activate the spindle brake
 - 1: Deactivate the spindle brake



Use thread-milling tools for cycle G800.



Danger of collision!

The hole depth must exceed the thread depth by at least $F/2$.

Thread milling in YZ plane G806

Starting with software version 625 952-05: G806 mills a thread in existing holes.

The cycle positions the tool on the end point of the thread within the hole. Then the tool approaches on "approaching radius R" and mills the thread. During this, the tool advances by the thread pitch F. Following that, the cycle retracts the tool and returns it to the starting point. With parameter V, you can program whether the thread is to be milled in one revolution or, with single-point tools, in several revolutions.

Parameters

- I Thread diameter
- X Starting point in X
- K Thread depth
- R Approach radius
- F Thread pitch
- J Direction of thread—(default: 0)
 - 0: Right-hand thread
 - 1: Left-hand thread
- H Cutting direction (default: 0)
 - 0: Up-cut milling
 - 1: Climb milling
- V One rotation / several rotations
 - 0: The thread is milled in a 360-degree rotation
 - 1: The thread is milled in several rotations (single-point tool)
- O Spindle brake (O is evaluated if the brake is entered in machine parameter 1019, ..)—default: 0
 - 0: Activate the spindle brake
 - 1: Deactivate the spindle brake

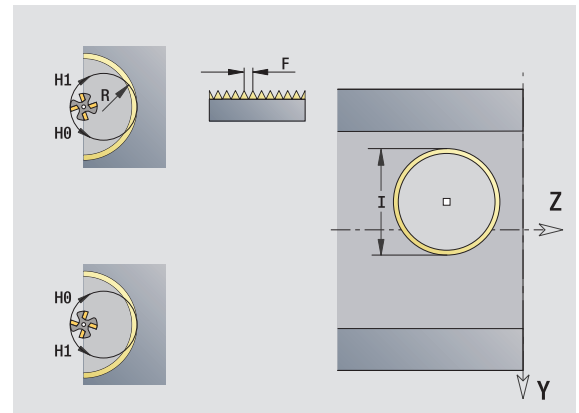
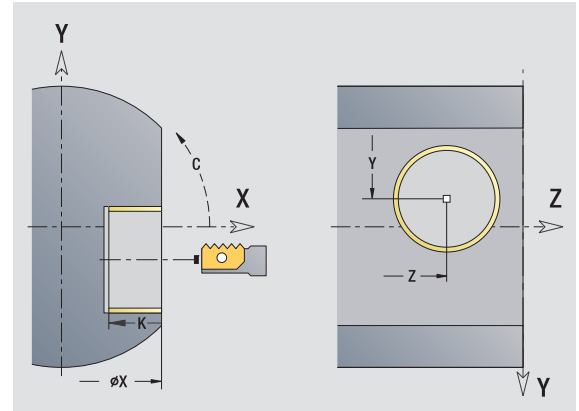


Use thread-milling tools for cycle G806.



Danger of collision!

The hole depth must exceed the thread depth by at least $F/2$.



Hobbing G808

Available as of software version 625 952-05.

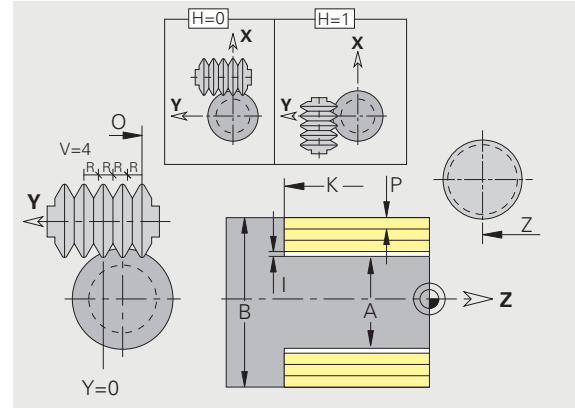
G808 mills a gear profile from the "starting point in Z" to the "end point K". In W you enter the angular position of the tool.

If an oversize has been programmed, hobbing is split up in rough-machining and subsequent finishing.

In parameters O, R and V you define the tool shift. Shifting by R ensures a uniform wear of the hob cutter.

Parameters

Z	Starting point
K	End point
A	Root circle diameter
B	Outside diameter
W	Angular position
J	Number of teeth, workpiece
D	Direction of rotation of the workpiece
	■ 3: M3
	■ 4: M4
S	Surface speed [m/min]
P	Maximum infeed
F	Feed per revolution
I	Oversize
E	Finishing feed rate
O	Shift starting position
R	Shift value
V	Number of shifts
H	Infeed axis
	■ 0: Tool infeed is performed in the X axis
	■ 1: Tool infeed is performed in the Y axis
Q	Workpiece spindle
	■ 0: Spindle no. 0 (main spindle) holds the workpiece
	■ 3: Spindle no. 3 (opposing spindle) holds the workpiece
U	Gear ratio of tool



1.12 Simulation

Simulation of the tilted plane

Contour graphics: The simulation displays the YZ view of the workpiece and the contours of the tilted planes in the **side view**. To represent the drilling patterns and milling contours perpendicularly to the tilted plane, i.e. without distortion, the simulation ignores the rotation of the coordinate system and a shift within the rotated coordinate system.

With contour graphics for tilted planes, please note the following:

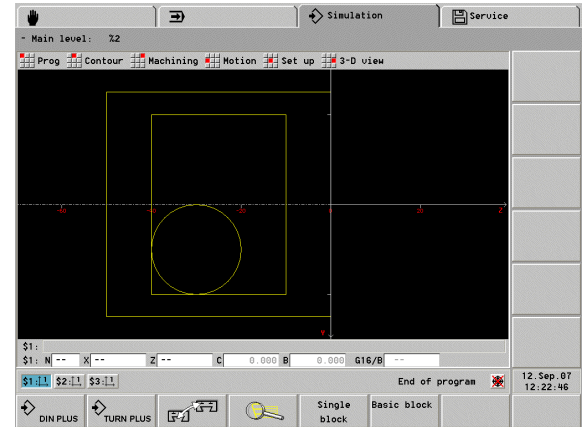
- The parameter “K” of G16 or SURFACE_Y defines the “start” of the drilling pattern or milling contour in the Z direction.
- The drilling patterns and milling contours are drawn perpendicularly to the tilted plane. This results in a “shift” relative to the turning contour.

The figure and the associated program section illustrate this situation.

Milling, drilling and boring operations: When you use the **side view** to display the tool paths in the tilted plane, the same rules apply as for the contour graphics.

When working in tilted planes, the **front window** shows the “outline” of the tool. The tool width is simulated true to scale. In this way, you can check the overlap of milling paths. The tool paths are also represented true to scale (in perspective view) as line graphics.

In all “additional windows,” the simulation shows the tool and the cutting path when the tool is perpendicular to the relevant plane. A tolerance of $\pm 5^\circ$ is taken into account. When the tool is not perpendicular to the plane, it is represented as a “light dot” and the tool path is depicted as a line.



Example: “Contour in tilted plane”

```

...
FINISHED PART
N2 G0 X0 Z0
N3 G1 X50
N4 G1 Z-50
N5 G1 X0
N6 G1 Z0
MANTEL_Y X50 C0 B80 I25 K-10 H0 [SURFACE_Y]
N7 G386 Z0 Ki10 B-30 X50 C0 [Single
surface]
MANTEL_Y X50 C0 B20 I25 K-20 H1 [SURFACE_Y]
N8 G384 Z-10 Y10 X50 R10 P5 [Full circle]
...

```

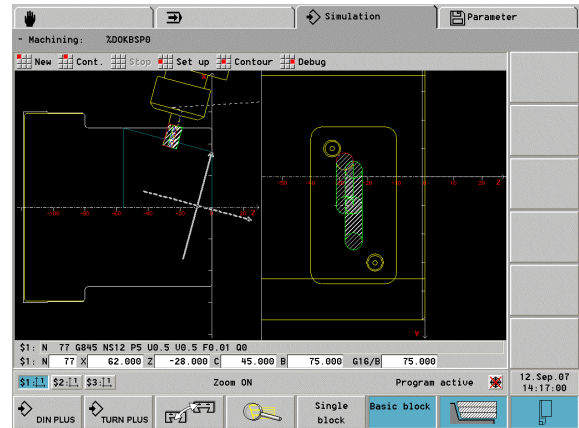
Displaying the coordinate system

The simulation can show the shifted/rotated coordinate system in the "lathe window," if required. To use this feature, you need to stop the simulation.



- Press the Continue key. The simulation displays the current coordinate system.

The coordinate system disappears when the next command is simulated or when you press Continue once again.



Position display with the B and Y axes

The following boxes of the display cannot be edited:

- **N**: Block number of the NC source block
- **X, Z, C**: Position values (actual values)

The other boxes can be set with the PgUp / PgDown keys or with the "Set > Status line" menu:

- Default settings (values of the selected slide):
 - **Y**: Position value (actual value)
 - **T**: Tool data with turret pocket, magazine pocket (in "(..)") and ID number
- Technology data settings:
 - Speed
 - Feed rate
 - Direction of spindle rotation
- B axis settings:
 - **B**: Tilt angle of the B axis
 - **G16/B**: Angle of the tilted plane

1.13 TURN PLUS: Tool Magazine and B Axis

Tool magazine

TURN PLUS recognizes whether a turret is used as the tool carrier or whether the tool carrier is loaded from a magazine. The tool selection depends on the setting of machining parameter 2:

- "Tools from turret" setting: TURN PLUS uses the tools entered in the magazine list.
- "Tools from database" setting: TURN PLUS looks for suitable tools in the database.
- "Combined" setting: TURN PLUS looks for suitable tools in both the magazine list and the database.

When a tool magazine is inserted, the tools that are used are listed in the PLATE MAGZN. section when TURN PLUS generates the NC program. TURN PLUS also automatically generates G714 for the tool change.

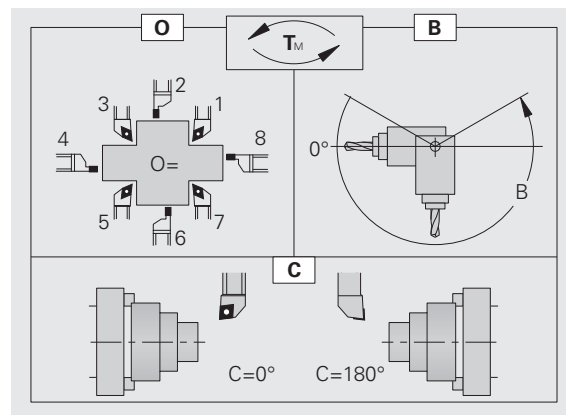
Tools for the B axis

By tilting the B axis and rotating the tool you can bring it into positions that enable you to use one and the same tool to machine in the longitudinal and transverse (or radial and axial) directions on the main and opposing spindles. TURN PLUS supports this flexible use of tools in the **IWG**.

After you have selected the tool, the IWG opens the "Tool position" dialog box. You can now specify the position of the tool for the next machining operation to be performed.

Parameters

- O Orientation of turning tools. Position of the tool's cutting edge (see figure).
- Tool positions 1, 3, 5, 7: For roughing, finishing and button tools (neutral tools are recognized by the tool angle).
 - Tool positions 2, 4, 6, 8: For recessing and threading cycles (a "right-hand" or "left-hand" tool is defined in the tool data).
- B B axis angle. Angle to which the B axis is tilted.
- C Offset position angle of turning tools.
- 0°: Tool attitude "normal"
 - 180°: Tool attitude "upside down"



1.14 TURN PLUS: Y Axis



The AWG can only be used for slides that have no B axis. If a slide with a B axis has been declared in the program head, automatic working plan generation is not possible.

Y axis - Basics

TURN PLUS supports the definition of milling contours and holes and creates working plans for milling and drilling with the Y axis.

Milling contours / holes: Milling contours are considered to be figures (slots, rectangles, etc.), linear or circular patterns, or “free” contours that you have defined. Holes are considered to be individual holes or hole patterns.

Milling, drilling and boring operations: TURN PLUS supports milling, drilling and boring in the IWG (Interactive Working plan Generation) and AWG (Automatic Working plan Generation).

Before you use the AWG, assign machining attributes to the contour (see User's Manual). With the familiar methods offered by TURN PLUS, you can determine the sequence of milling operations, the use of tools, etc.



Note on data input: When working with hierarchical milling contours (pocket inside pocket, holes/figures on a surface, etc.), you can use the up/down cursor keys to select among all the contours provided on the hierarchy level of a “workpiece surface.” With the right/left cursor keys you can select among the milling contours of a hierarchy level.

Definition of milling contours

Before programming a milling contour, define the workpiece blank and the turning contour.

“Free” contour: With the elements “line” and “arc,” you can define any contour. You simply define the “starting point of contour,” describe the contour, and then define the depth of the pocket/contour.

Programming a “free” contour

- ▶ Select “Workpiece > Finished part > Figure > Contour.”
- ▶ If not yet defined: Select the input plane (XY front, XYR back, ZY lateral surface).
- ▶ Set the reference element.
- ▶ Check the “reference data” and complete them, if necessary.
- ▶ Define the contour
- ▶ Check the contour in the control graphic.

Programming figures/patterns

- ▶ Select “Workpiece > Finished part > Pattern” (or “.. > Figure”).
- ▶ Select the pattern or figure.
- ▶ If not yet defined: Select the input plane (XY front, XYR back, ZY lateral surface).
- ▶ Set the reference element.
- ▶ Check the “reference data” and complete them, if necessary.
- ▶ Enter the parameters for the pattern/figure.
- ▶ Check the pattern/figure in the control graphic.



- If you use the main view for programming, TURN PLUS will ask you to set the working plane. If you use the front, rear or surface window, the control will assume this plane to be the working plane.
- You can change the “active window” with the PgUp/PgDn keys or the cursor.

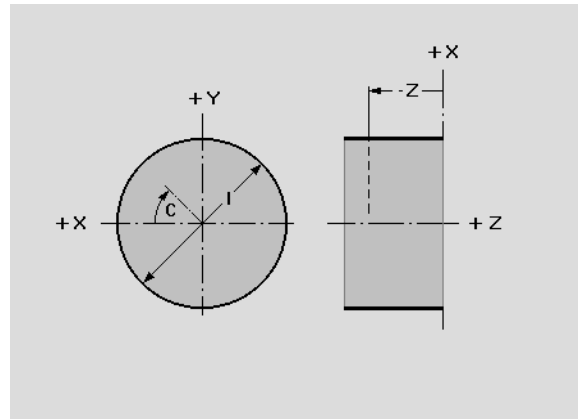
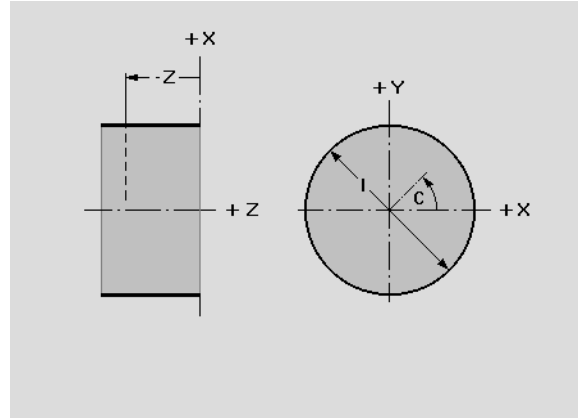
1.15 TURN PLUS: XY Plane Contours

Reference data - XY front/XYR back

In the "reference data," you can define the position of the plane.

Parameters

- C Spindle angle (spindle position); (default: 0)
- I Limit diameter. Serves to limit the cutting area if the figure lies partly outside the workpiece.
- Z Reference dimension. Position of reference plane.

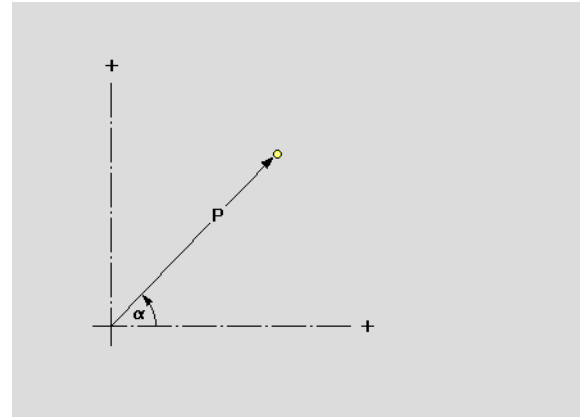
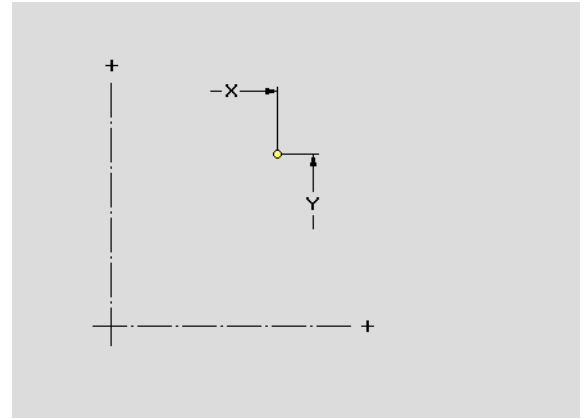


XY plane: Starting point of contour

This function defines the starting point in the XY plane.

Parameters

- X Starting point of contour (radius)
- Y Starting point of contour
- P Starting point of the contour in polar coordinates
- a Starting point of contour in polar coordinates (reference angle: positive X axis)

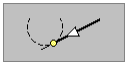


XY plane: Linear element

This function defines a line segment in the XY plane.

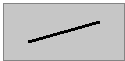
Parameters

- X End point in Cartesian coordinates (radius)
- Y End point in Cartesian coordinates
- Xi Incremental end point
- Yi Incremental end point
- P End point in polar coordinates
- a End point in polar coordinates (reference angle: positive X axis)
- W Angle of the line (for reference see illustration)
- WV Counterclockwise angle to the preceding element. If the preceding element is an arc: Angle to the tangent.
- WN Counterclockwise angle to the following element. If the following element is an arc: Angle to the tangent.
- L Length of element



Tangential/nontangential: Specify the transition to the next contour element

Defining a linear element:

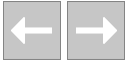


Call the lines menu.

Select the line direction.



Vertical line



Horizontal line



Line at angle

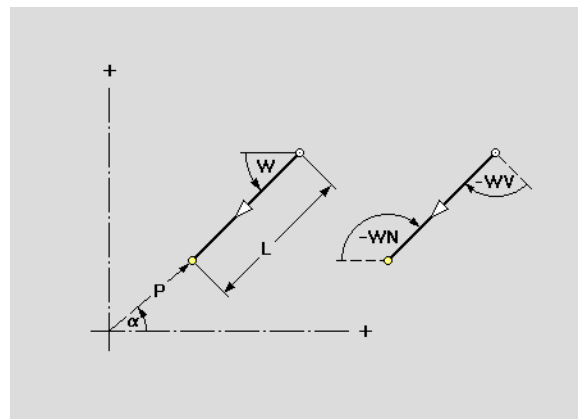
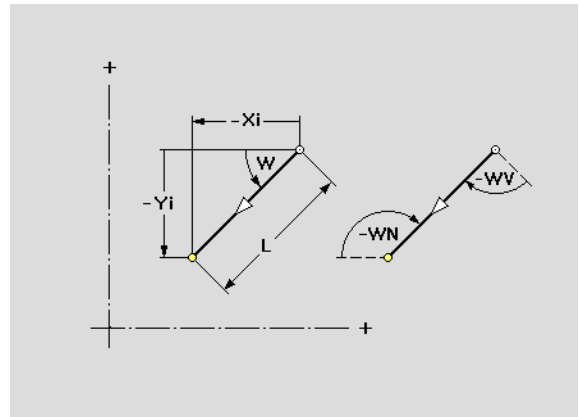
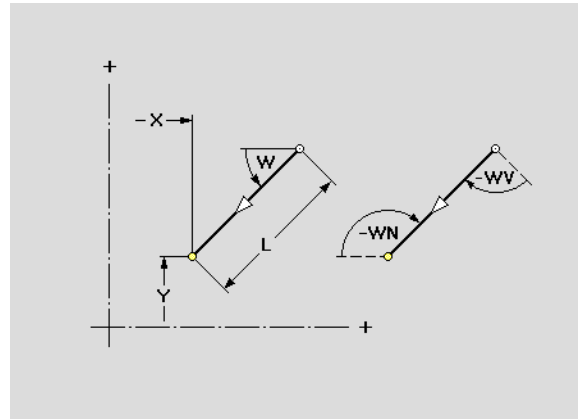


Line at angle



Line in any direction

Enter the line dimensions and define the transition to the next element.



XY plane: Arc

This function defines a circular element in the XY plane.

Parameters

End point of the arc

- X End point in Cartesian coordinates (radius)
- Y End point in Cartesian coordinates
- Xi Incremental end point
- Yi Incremental end point
- P End point in polar coordinates
- a End point in polar coordinates (reference angle: positive X axis)
- Pi End point polar, incremental (linear distance from starting point to end point)
- ai End point polar, incremental (reference: Angle between an imaginary line intersecting the starting point and parallel to the X axis, and another line from the starting point to the end point)

Center point of the arc

- XM Center point in Cartesian coordinates (radius)
- YM Center point in Cartesian coordinates
- XMi Incremental center point
- YMi Incremental center point
- PM Center point in polar coordinates
- b Center point in polar coordinates (reference angle: positive X axis)
- PMi Center point polar, incremental (linear distance from starting point to end point)
- bi Center point polar, incremental (reference: Angle between an imaginary line intersecting the starting point and parallel to the X axis, and another line from the starting point to the end point)

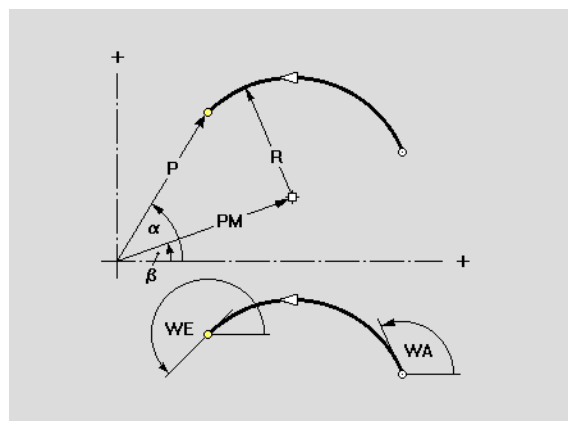
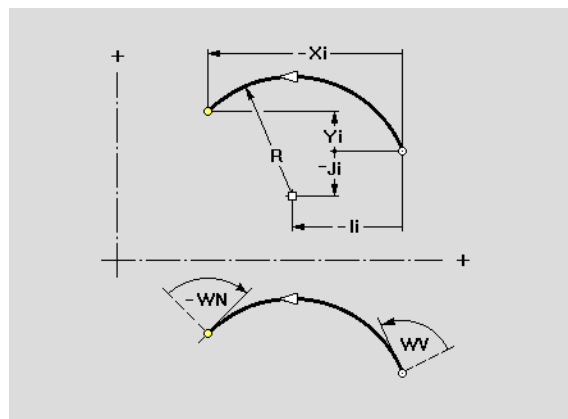
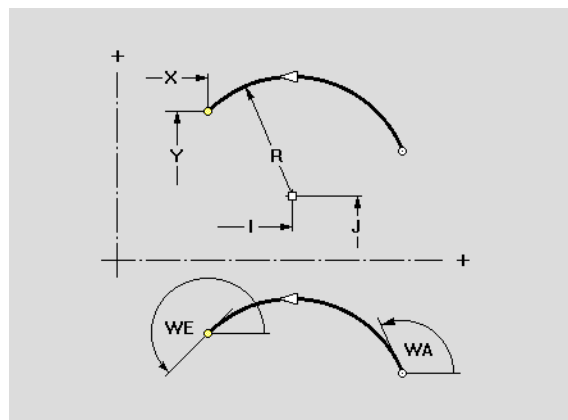
Other parameters

- R Arc radius



Tangential/nontangential: Specify the transition to the next contour element

- WA Angle between positive X axis and tangent in starting point of arc
- WE Angle between positive X axis and tangent in end point of arc
- WV Counterclockwise angle between preceding element and tangent in starting point of arc. If the preceding element is an arc: Angle to the tangent.
- WN Counterclockwise angle between tangent in arc end point and following element. If the following element is an arc: Angle to the tangent.



Defining a circular element:

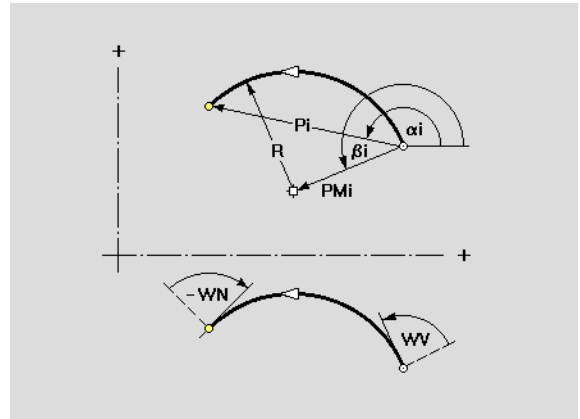


Call the arcs menu.



Select the direction of rotation.

Enter the arc dimensions and define the transition to the next element.



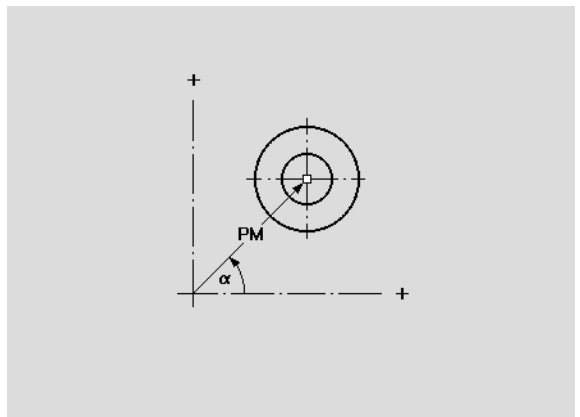
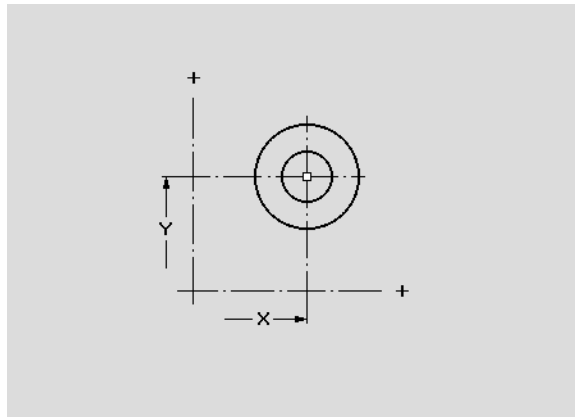
XY plane: Single hole

This function defines a single hole in the XY plane. The hole may contain the following elements:

- Centering
- Core hole
- Countersinking
- Thread

Hole reference point parameters

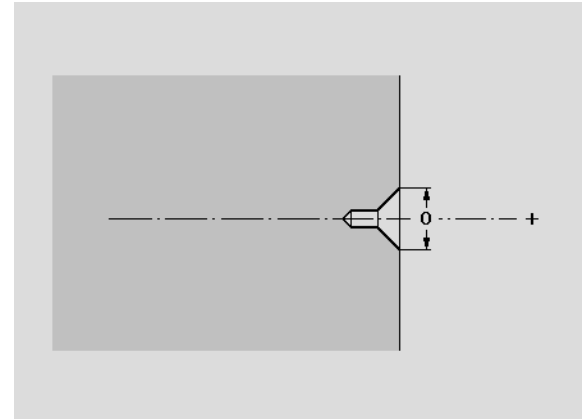
- X Position—center point in Cartesian coordinates (radius)
- Y Position—center point in Cartesian coordinates
- PM Position—center point in polar coordinates
- a Position—center point in polar coordinates (reference angle: positive X axis)



Centering in XY plane

Centering parameters

Q Centering diameter



Core hole in XY plane

Core hole parameters

B Diameter of hole

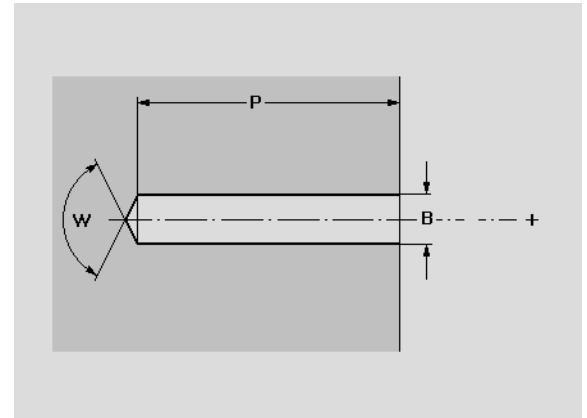
P Depth of hole (excluding point)

W Point angle

■ $W=0^\circ$: the AWG generates a "feed rate reduction ($V=1$)" for the drilling cycle

■ $W>0^\circ$: point angle

Fit: H6 to H13 or "without fit"



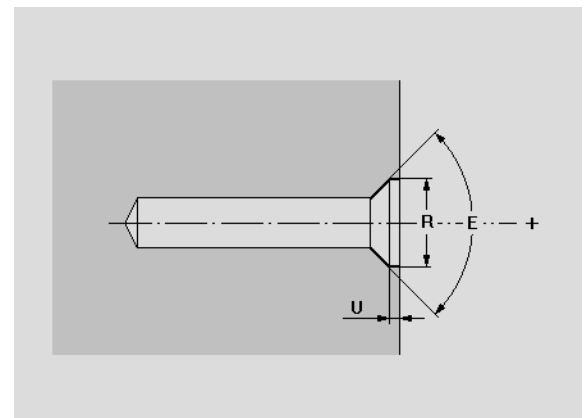
Countersinking in XY plane

Countersinking parameters

R Sinking diameter

U Sinking depth

E Sinking angle



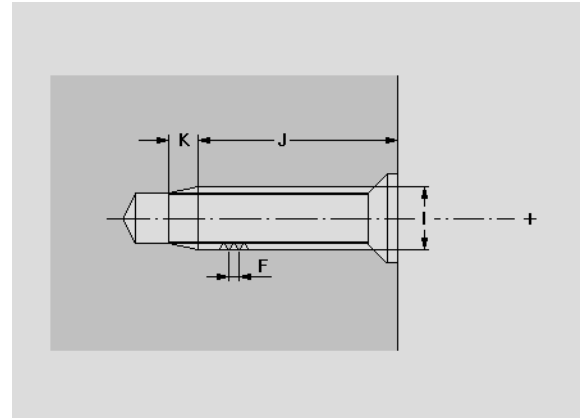
Tapping in XY plane

Thread parameters

- I Nominal diameter
- J Thread depth
- K Thread runout length
- F Thread pitch

Type of thread:

- Right-hand thread
- Left-hand thread

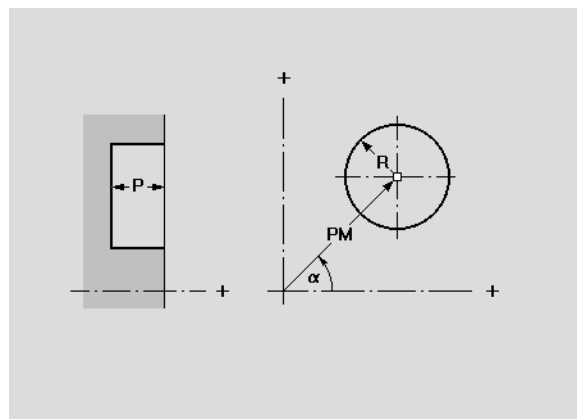
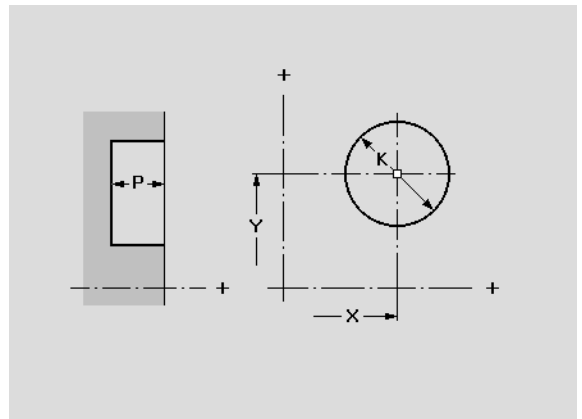


XY plane: Circle (full circle)

This function defines a full circle in the XY plane.

Parameters

- X Center point in Cartesian coordinates (radius)
- Y Center in Cartesian coordinates
- PM Center point in polar coordinates
- a Center point in polar coordinates (reference angle: positive X axis)
- R Circle radius
- K Circle diameter
- P Depth of the figure

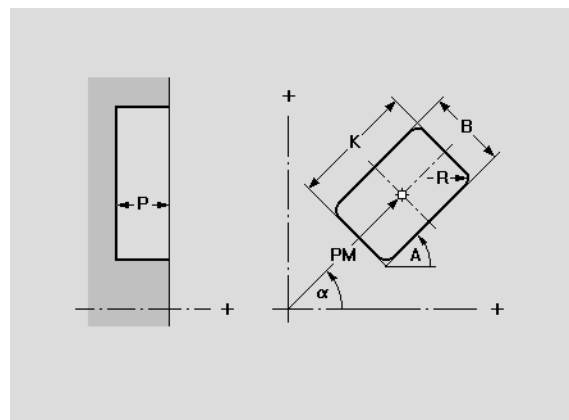
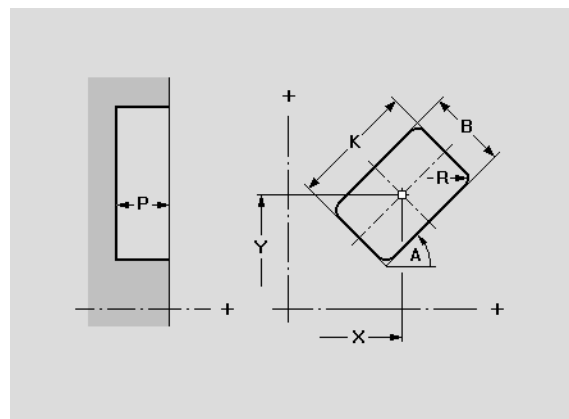


XY plane: Rectangle

This function defines a rectangle in the XY plane.

Parameters

- X Center point in Cartesian coordinates (radius)
- Y Center in Cartesian coordinates
- PM Center point in polar coordinates
- a Center point in polar coordinates (reference angle: positive X axis)
- A Position angle (reference: positive X axis and long side of rectangle)
- K Length of rectangle
- B Width of rectangle
- R Chamfer/rounding
 - Width of chamfer
 - Radius of rounding
- P Depth of the figure

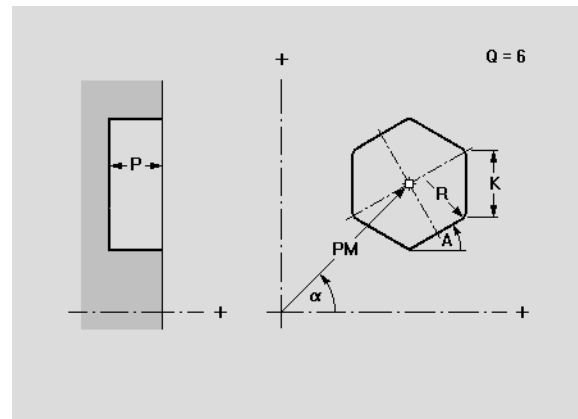
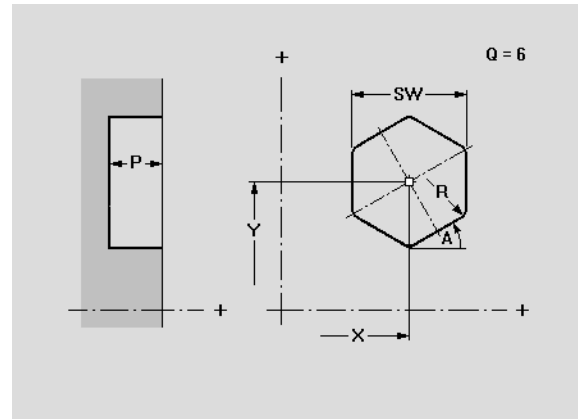


XY plane: Polygon

This function defines a polygon in the XY plane.

Parameters

- X Center point in Cartesian coordinates (radius)
- Y Center in Cartesian coordinates
- PM Center point in polar coordinates
- a Center point in polar coordinates (reference angle: positive X axis)
- A Angle to a polygon side (reference: X axis)
- Q Number of corners ($Q \geq 3$)
- K Edge length
- SW Width across flats (inscribed circle diameter)
- R Chamfer/rounding
 - Width of chamfer
 - Radius of rounding
- P Depth of the figure

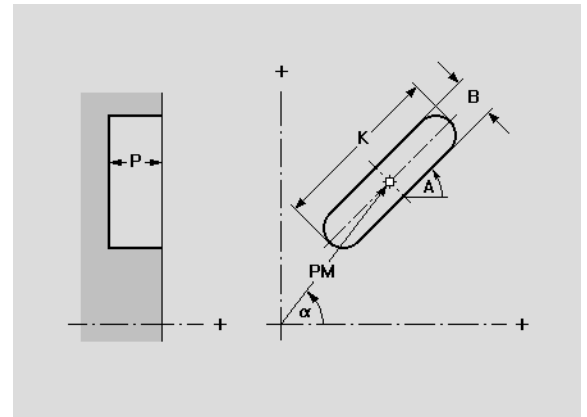
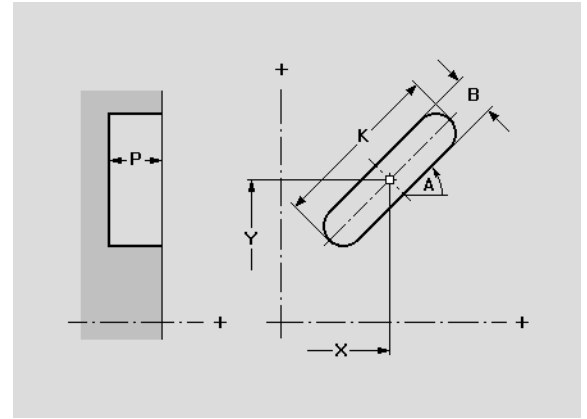


XY plane: Linear slot

This function defines a linear slot in the XY plane.

Parameters

X	Center point in Cartesian coordinates (radius)
Y	Center in Cartesian coordinates
PM	Center point in polar coordinates
a	Center point in polar coordinates (reference angle: positive X axis)
A	Angle to longitudinal axis of slot (reference: X axis)
K	Slot length
B	Slot width
P	Depth of the figure

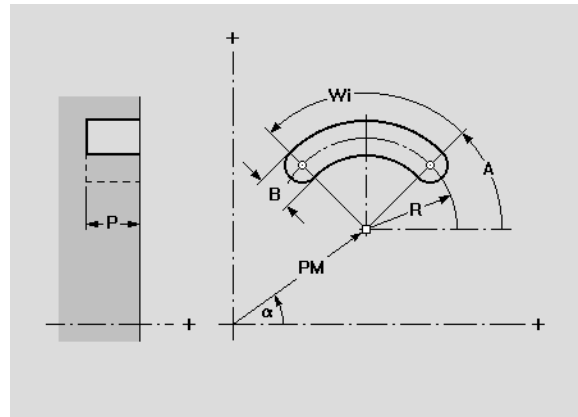
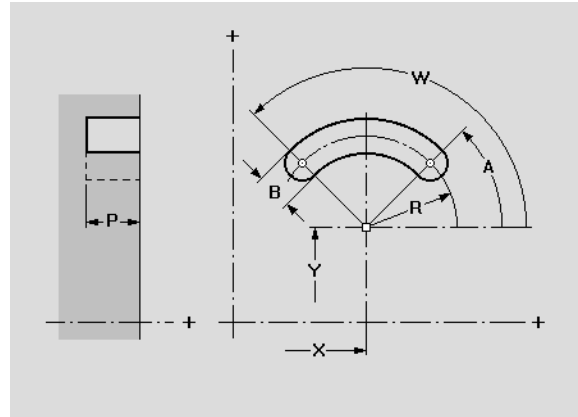


XY plane: Circular slot

This function defines a circular slot in the XY plane.

Parameters

X	Center of curvature in Cartesian coordinates (radius)
Y	Center of curvature in Cartesian coordinates
PM	Center of curvature in polar coordinates
a	Center of curvature in polar coordinates (reference angle: positive X axis)
A	Starting angle of slot (reference: X axis)
W	End angle of slot (reference: X axis)
R	Curvature radius (reference: center point path of the slot)
B	Slot width
P	Depth of the figure

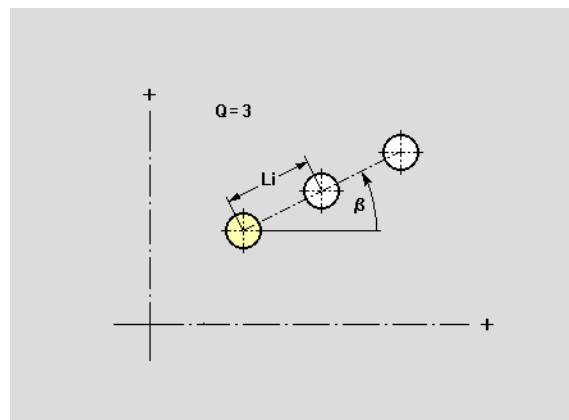
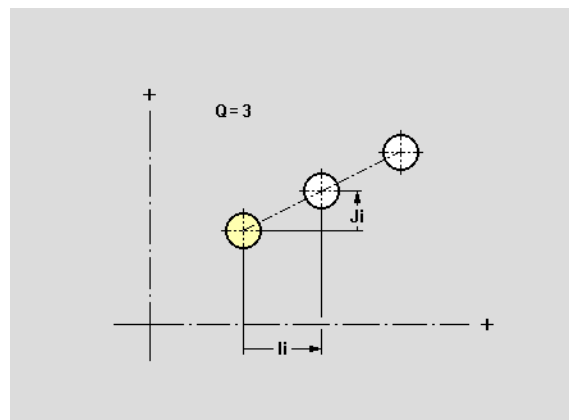
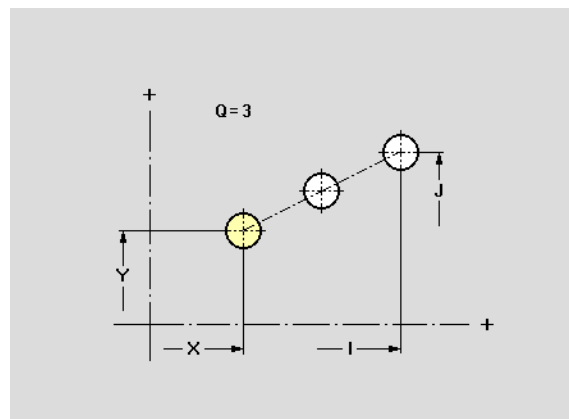


XY plane: Linear drilling pattern

This function defines a linear drilling pattern in the XY plane.

Parameters

- X Starting point of pattern in Cartesian coordinates (radius)
 - Y Starting point of pattern in Cartesian coordinates
 - a Starting point of pattern in polar coordinates (reference angle: positive X axis)
 - P Starting point of pattern in polar coordinates
 - Q Number of holes
 - I End point of pattern in Cartesian coordinates (radius)
 - J End point of pattern in Cartesian coordinates
 - li Distance in X direction between two holes
 - Ji Distance in Y direction between two holes
 - b Angle to longitudinal axis of pattern (reference: X axis)
 - L Total length of pattern
 - Li Distance between two holes (pattern distance)
- Description of the hole (see "XY plane: Single hole" on page 77)



XY plane: Circular drilling pattern

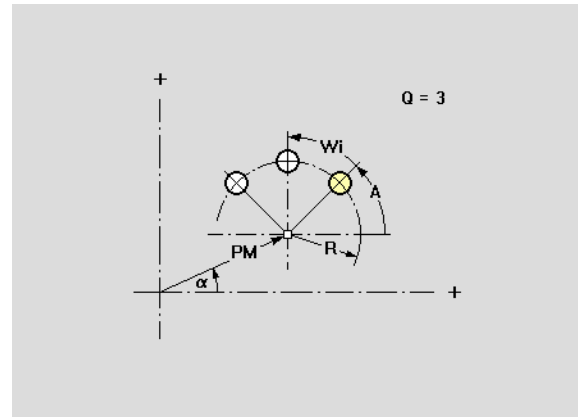
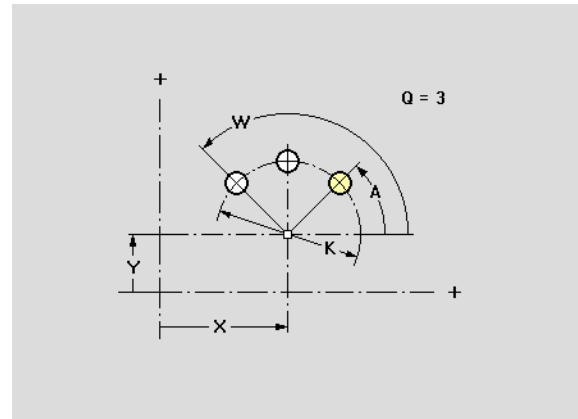
This function defines a circular drilling pattern in the XY plane.

Parameters

- X Center of pattern in Cartesian coordinates (radius)
- Y Center of pattern in Cartesian coordinates
- a Center of pattern in polar coordinates (reference angle: positive X axis)
- PM Center of pattern in polar coordinates
- Q Number of holes/figures
- Orientation:
- Clockwise
 - Counterclockwise
- R Radius of the pattern
- K Diameter of the pattern
- A Starting angle, position of the first hole (reference: X axis)
- W End angle, position of the last hole (reference: X axis)
- Wi Angle between two holes (algebraic sign has no effect)
- Description of the hole (see "XY plane: Single hole" on page 77)

Special cases of starting and end angles (A, W):

- Without A and W: Figures are arranged on a full circle, starting at 0°
- Without W: Figures are arranged on a full circle

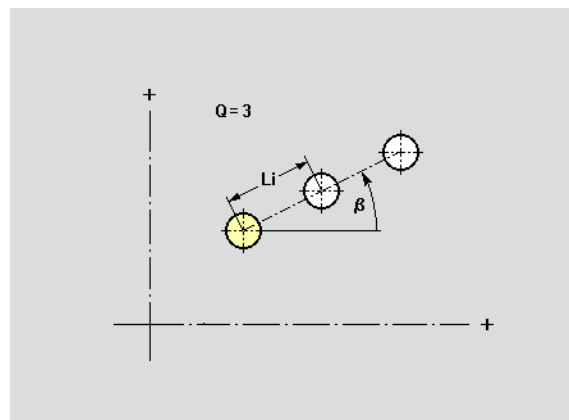
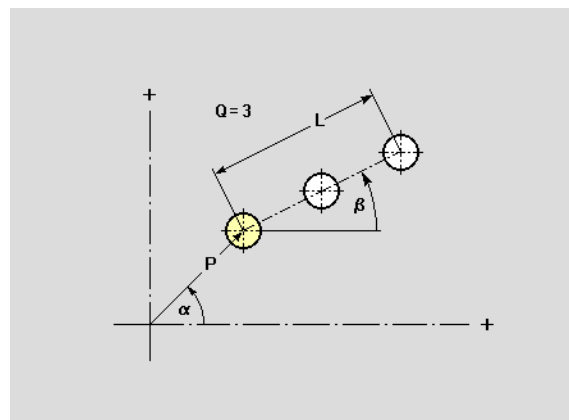
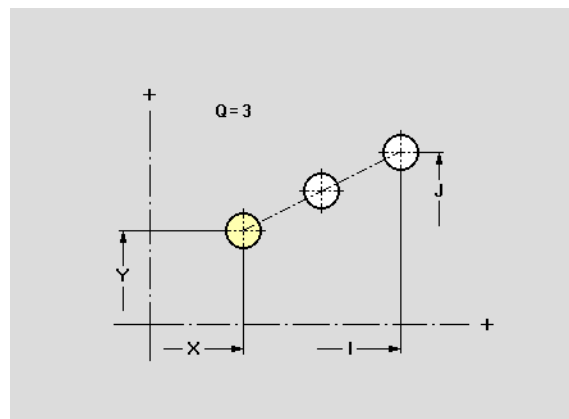


XY plane: Linear figure pattern

This function defines a linear figure pattern in the XY plane.

Parameters

X	Starting point of pattern in Cartesian coordinates (radius)
Y	Starting point of pattern in Cartesian coordinates
a	Starting point of pattern in polar coordinates (reference angle: positive X axis)
P	Starting point of pattern in polar coordinates
Q	Number of figures
I	End point of pattern in Cartesian coordinates (radius)
J	End point of pattern in Cartesian coordinates
li	Distance in X direction between two figures
Ji	Distance in Y direction between two figures
b	Angle to longitudinal axis of pattern (reference: X axis)
L	Total length of pattern
Li	Distance between two figures (pattern distance)
	Description of the figure



XY plane: Circular figure pattern

This function defines a circular figure pattern in the XY plane.

Parameters

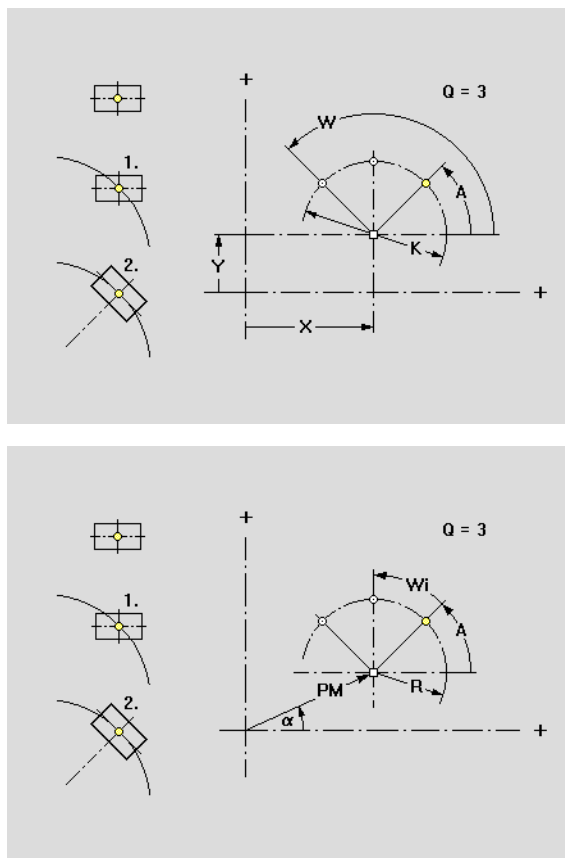
- X Center of pattern in Cartesian coordinates (radius)
- Y Center of pattern in Cartesian coordinates
- a Center of pattern in polar coordinates (reference angle: positive X axis)
- PM Center of pattern in polar coordinates
- Q Number of figures
- Orientation:
- Clockwise
 - Counterclockwise
- R Radius of the pattern
- K Diameter of the pattern
- A Starting angle, position of the first figure (reference: X axis)
- W End angle, position of the last figure (reference: X axis)
- Wi Angle between two figures (algebraic sign has no effect)
- Position of the figures
- Normal position: The original figure is rotated about the center of the pattern (rotation)
 - Original position: The position of the original figure is maintained (translation)
- Description of the figure

Special cases of starting and end angles (A, W):

- Without A and W: Figures are arranged on a full circle, starting at 0°
- Without W: Figures are arranged on a full circle



When defining patterns with circular slots, the “center of curvature” is added to the pattern position.

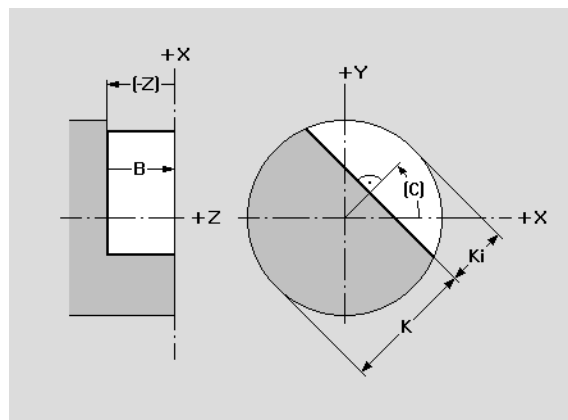


XY plane: Single surface

This function defines a single surface in the XY plane.

Parameters

- Ki Depth (of material to be milled)
- K Residual depth (remaining material)
- B Width (reference: reference edge Z)
 - $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction

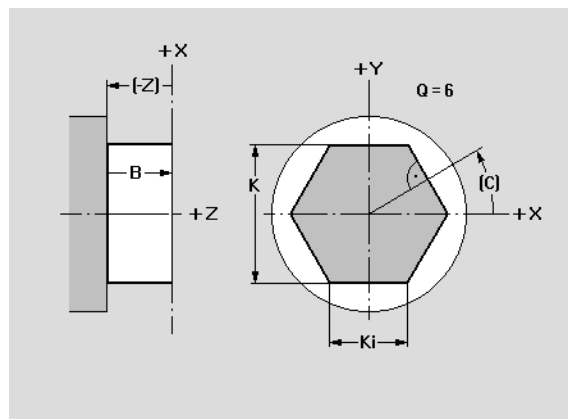


XY plane: Centric polygon

This function defines polygonal surfaces in the XY plane.

Parameters

- Q Number of sides ($Q \geq 2$)
- K Width across flats (inscribed circle diameter)
- Ki Edge length
- B Width (reference: reference edge Z)
 - $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction



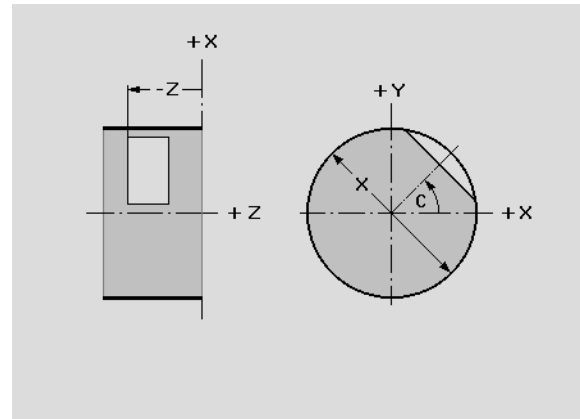
1.16 TURN PLUS: YZ Plane Contours

Reference data - Y lateral surface

In the "reference data," you can define the position of the plane.

Parameters

- C Spindle angle (spindle position); (default: 0)
- Z Limit dimension—reference position for single surfaces and centric polygons
- X Reference diameter
 - Reference position for the figures/contours
 - Serves to limit the cutting area if the figure lies partly outside the workpiece

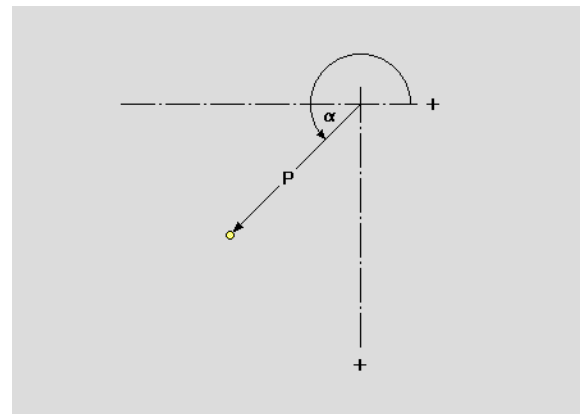
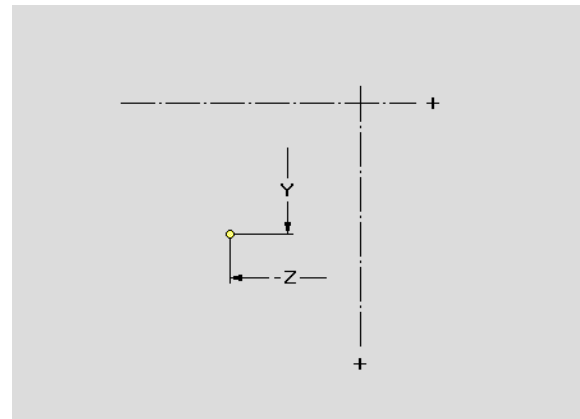


YZ plane: Starting point of contour

This function defines the starting point in the YZ plane.

Parameters

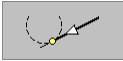
- Y Starting point of the contour in Cartesian coordinates
- Z Starting point of the contour in Cartesian coordinates
- P Starting point of the contour in polar coordinates
- a Starting point of the contour in polar coordinates (reference angle: positive Z axis)



YZ plane: Linear element

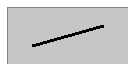
This function defines a line segment in the YZ plane.

Parameters

- Y End point in Cartesian coordinates
 - Z End point in Cartesian coordinates
 - Yi Incremental end point
 - Zi Incremental end point
 - P End point in polar coordinates
 - a End point in polar coordinates (reference angle: positive Z axis)
 - W Angle of the line (for reference see illustration)
 - WV Counterclockwise angle to the preceding element. If the preceding element is an arc: Angle to the tangent.
 - WN Counterclockwise angle to the following element. If the following element is an arc: Angle to the tangent.
 - L Length of element
- 

Tangential/nontangential: Specify the transition to the next contour element

Defining a linear element:



Call the lines menu.

Select the line direction.



Vertical line



Horizontal line



Line at angle

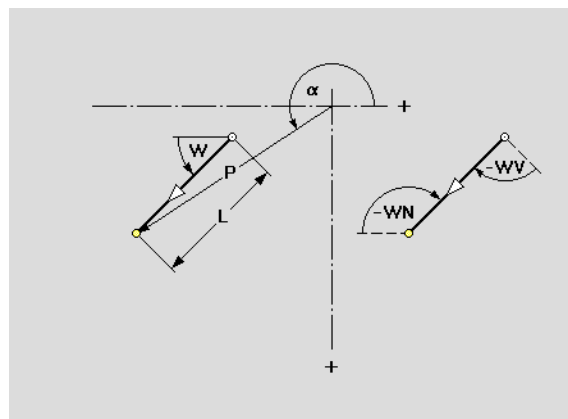
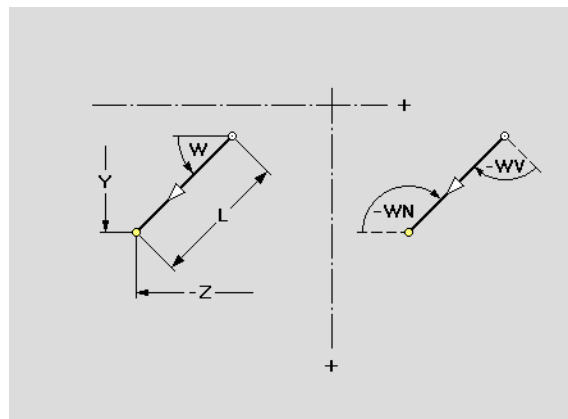


Line at angle



Line in any direction

Enter the line dimensions and define the transition to the next element.



YZ plane: Arc

This function defines a circular element in the YZ plane.

Parameters

End point of the arc

- Y End point in Cartesian coordinates
- Z End point in Cartesian coordinates
- Yi Incremental end point
- Zi Incremental end point
- P End point in polar coordinates
- a End point in polar coordinates (reference angle: positive Z axis)
- Pi End point polar, incremental (linear distance from starting point to end point)
- ai End point polar, incremental (reference: Angle between an imaginary line intersecting the starting point and parallel to the Z axis, and another line from the starting point to the end point)

Center point of the arc

- YM Center point in Cartesian coordinates (radius)
- ZM Center point in Cartesian coordinates
- YMi Incremental center point
- ZMi Incremental center point
- PM Center point in polar coordinates
- b Center point in polar coordinates (reference angle: positive Z axis)
- PMi Center point polar, incremental (linear distance from starting point to end point)
- bi Center point polar, incremental (reference: Angle between an imaginary line intersecting the starting point and parallel to the Z axis, and another line from the starting point to the end point)

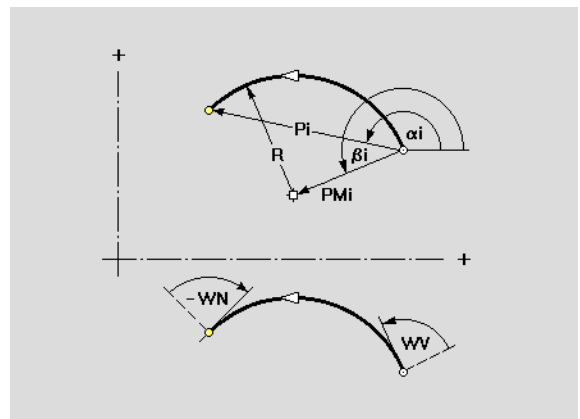
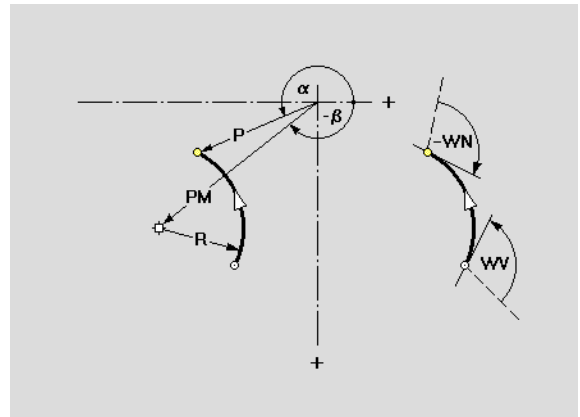
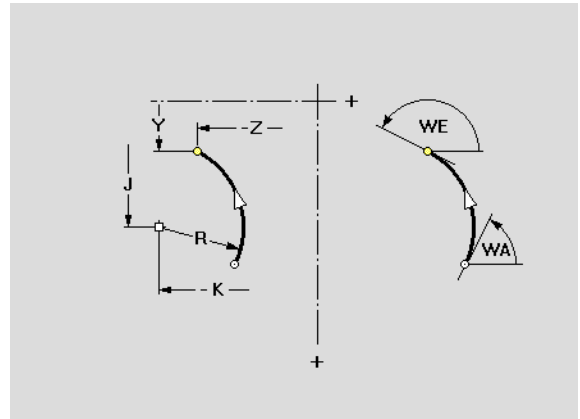
Other parameters

- R Arc radius



Tangential/nontangential: Specify the transition to the next contour element

- WA Angle between positive Z axis and tangent in starting point of arc
- WE Angle between positive Z-axis and tangent in end point of arc
- WV Counterclockwise angle between preceding element and tangent in starting point of arc. If the preceding element is an arc: Angle to the tangent.
- WN Counterclockwise angle between tangent in arc end point and following element. If the following element is an arc: Angle to the tangent.



Defining a circular element:

Call the arcs menu.



Select the direction of rotation.

Enter the arc dimensions and define the transition to the next element.

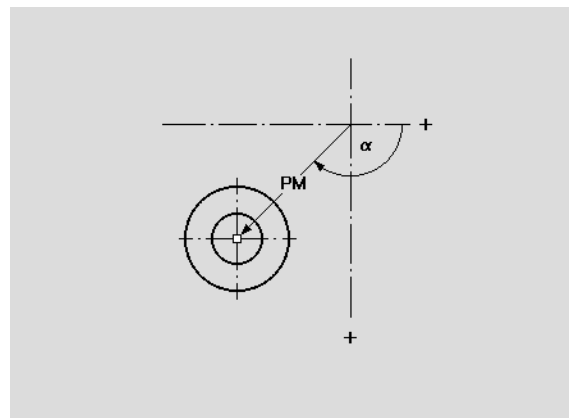
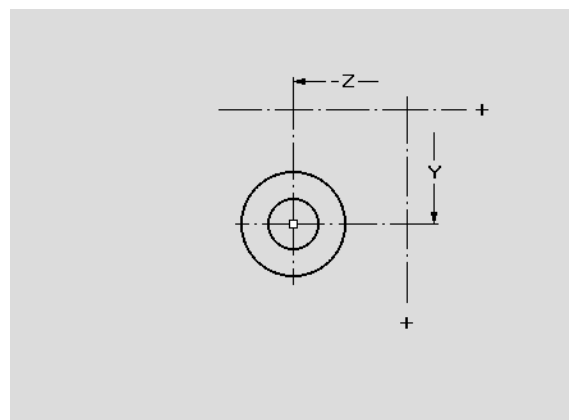
YZ plane: Single hole

This function defines a single hole in the YZ plane. The hole may contain the following elements:

- Centering
- Core hole
- Countersinking
- Threads

Hole reference point parameters

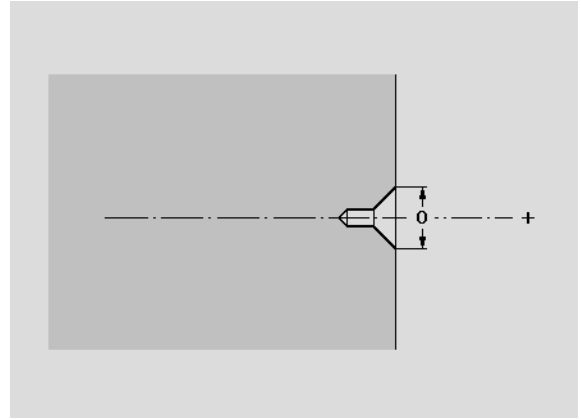
- Y Position - center point in Cartesian coordinates
- Z Position - center point in Cartesian coordinates
- PM Position - center point in polar coordinates
- a Position - center point in polar coordinates (reference angle: positive Z axis)



Centering in XY plane

Centering parameters

Q Centering diameter



Core hole in XY plane

Core hole parameters

B Diameter of hole

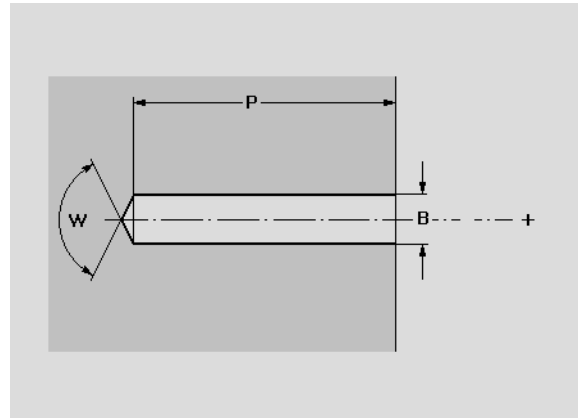
P Depth of hole (excluding point)

W Point angle

■ $W=0^\circ$: the AWG generates a "feed rate reduction ($V=1$)" for the drilling cycle

■ $W>0^\circ$: point angle

Fit: H6 to H13 or "without fit"



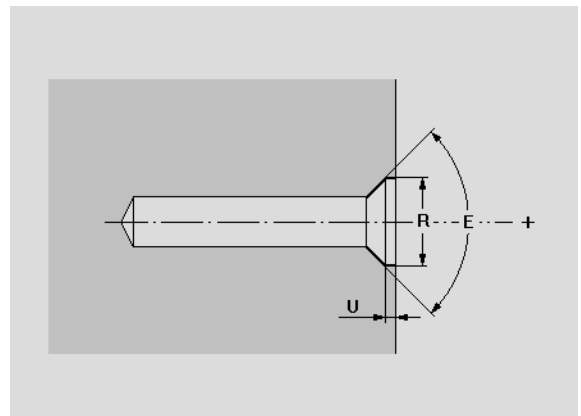
Countersinking in XY plane

Countersinking parameters

R Sinking diameter

U Sinking depth

E Sinking angle



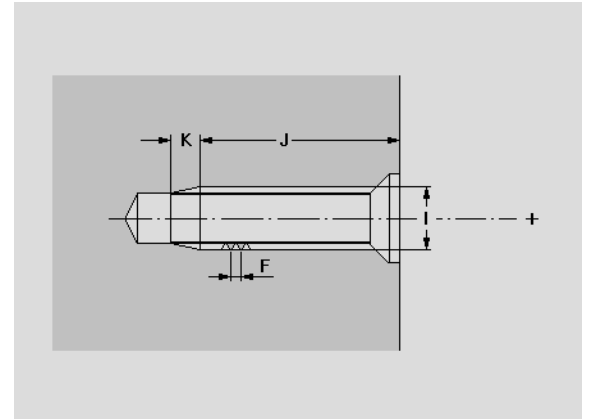
Tapping in XY plane

Thread parameters

- I Nominal diameter
- J Thread depth
- K Thread runout length
- F Thread pitch

Type of thread:

- Right-hand thread
- Left-hand thread

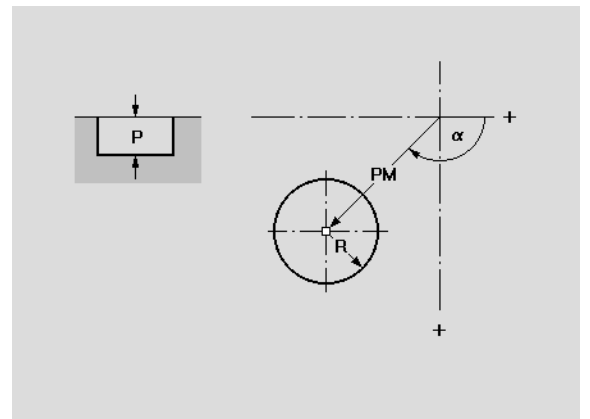
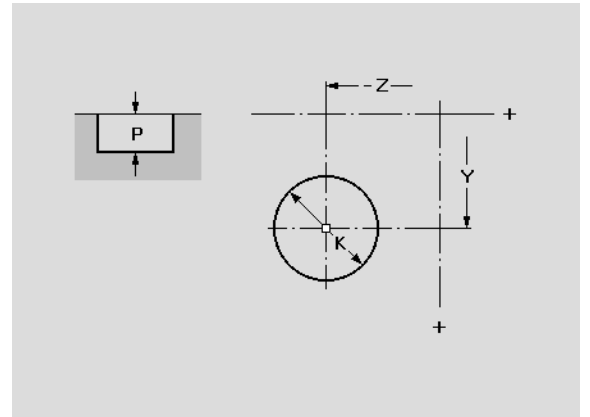


YZ plane: Circle (full circle)

This function defines a full circle in the YZ plane.

Parameters

- Y Center point in Cartesian coordinates
- Z Center in Cartesian coordinates
- PM Center point in polar coordinates
- a Center point in polar coordinates (reference angle: positive Z axis)
- R Circle radius
- K Circle diameter
- P Depth of the figure

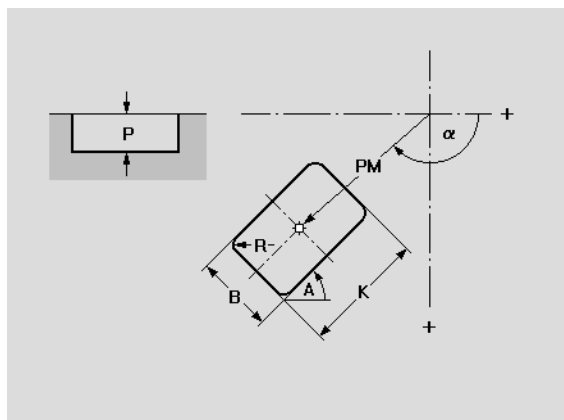
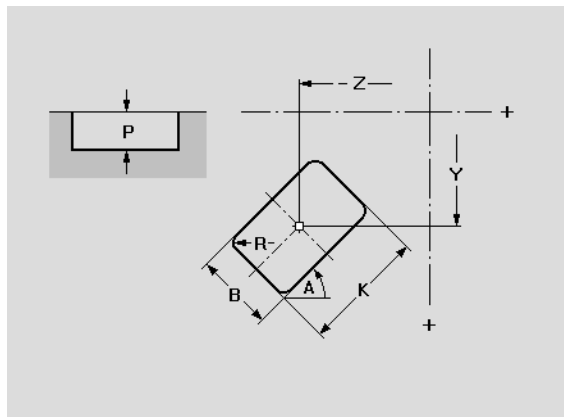


YZ plane: Rectangle

This function defines a rectangle in the YZ plane.

Parameters

- Y Center point in Cartesian coordinates
- Z Center in Cartesian coordinates
- PM Center point in polar coordinates
- a Center point in polar coordinates (reference angle: positive Z axis)
- A Position angle (reference: positive Z axis and long side of rectangle)
- K Length of rectangle
- B Width of rectangle
- R Chamfer/rounding
 - Width of chamfer
 - Radius of rounding
- P Depth of the figure

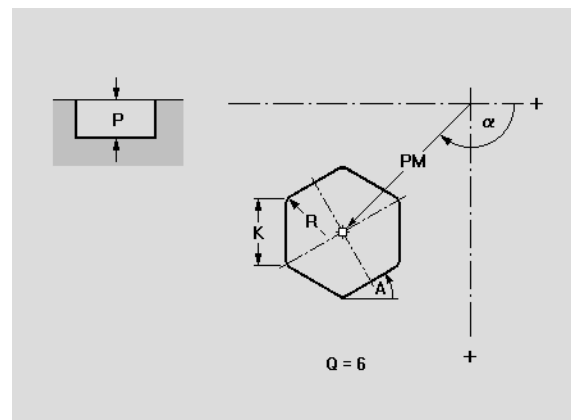
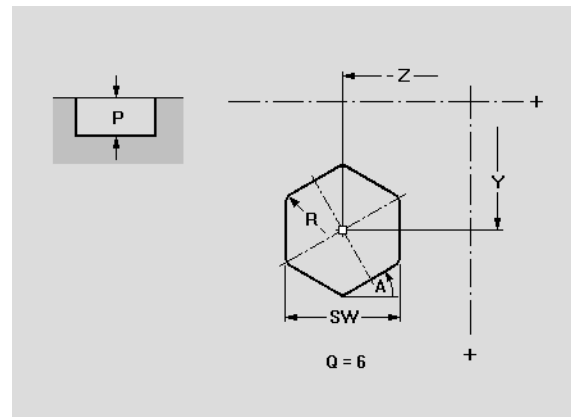


YZ plane: Polygon

This function defines a polygon in the YZ plane.

Parameters

- Y Center point in Cartesian coordinates
- Z Center in Cartesian coordinates
- PM Center point in polar coordinates
- a Center point in polar coordinates (reference angle: positive Z axis)
- A Angle to a polygon side (reference: Z axis)
- Q Number of corners ($Q \geq 3$)
- K Edge length
- SW Width across flats (inscribed circle diameter)
- R Chamfer/rounding
 - Width of chamfer
 - Radius of rounding
- P Depth of the figure

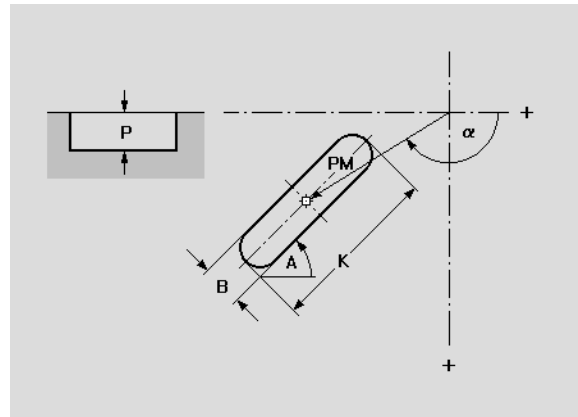
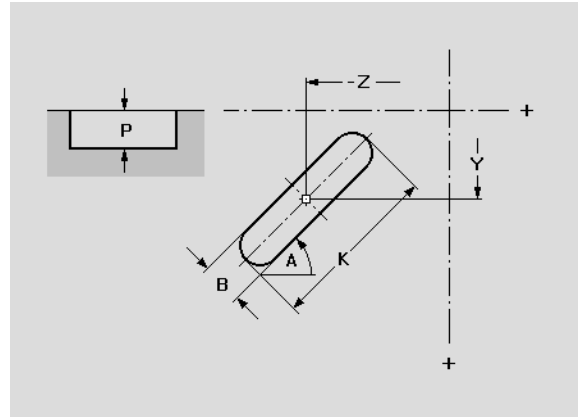


YZ plane: Linear slot

This function defines a linear slot in the YZ plane.

Parameters

Y	Center point in Cartesian coordinates
Z	Center in Cartesian coordinates
PM	Center point in polar coordinates
a	Center point in polar coordinates (reference angle: positive Z axis)
A	Angle to longitudinal axis of slot (reference: Z axis)
K	Slot length
B	Slot width
P	Depth of the figure

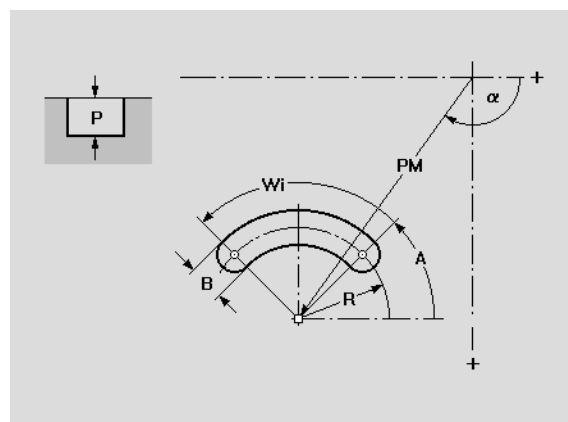
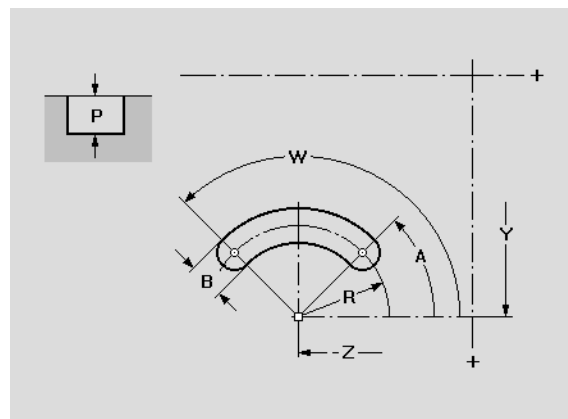


YZ plane: Circular slot

This function defines a circular slot in the YZ plane.

Parameters

Y	Center of curvature in Cartesian coordinates
Z	Center of curvature in Cartesian coordinates
PM	Center of curvature in polar coordinates
a	Center of curvature in polar coordinates (reference angle: positive Z axis)
A	Starting angle of slot (reference: Z axis)
W	End angle of slot (reference: Z axis)
R	Curvature radius (reference: center point path of the slot)
B	Slot width
P	Depth of the figure

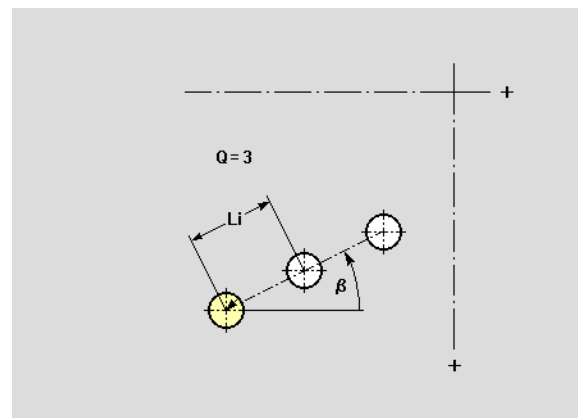
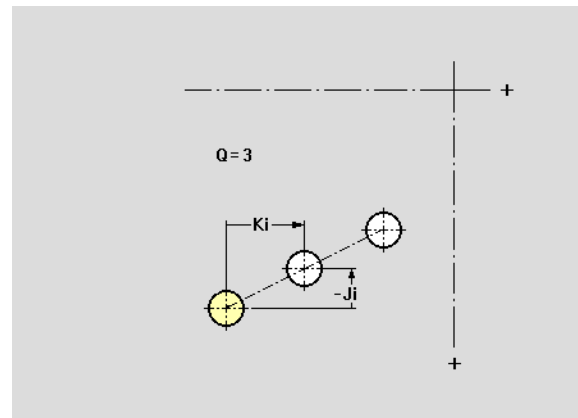
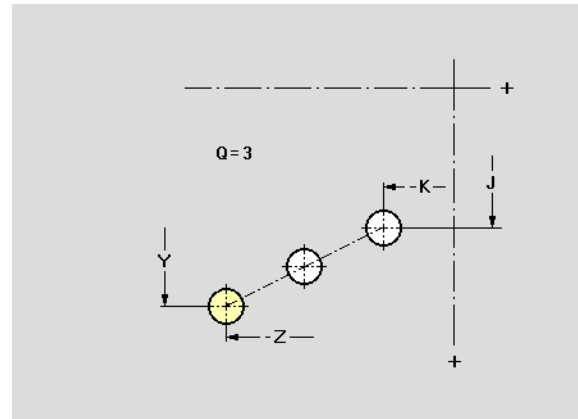


YZ plane: Linear drilling pattern

This function defines a linear drilling pattern in the YZ plane.

Parameters

- Y Starting point of pattern in Cartesian coordinates
 - Z Starting point of pattern in Cartesian coordinates
 - a Starting point of pattern in polar coordinates (reference angle: positive Z axis)
 - P Starting point of pattern in polar coordinates
 - Q Number of holes
 - J End point of pattern in Cartesian coordinates
 - K End point of pattern in Cartesian coordinates
 - Ji Distance in Y direction between two holes
 - Ki Distance in Z direction between two holes
 - b Angle to longitudinal axis of pattern (reference: Z axis)
 - L Total length of pattern
 - Li Distance between two holes (pattern distance)
- Description of the hole (see "XY plane: Single hole" on page 77)



YZ plane: Circular drilling pattern

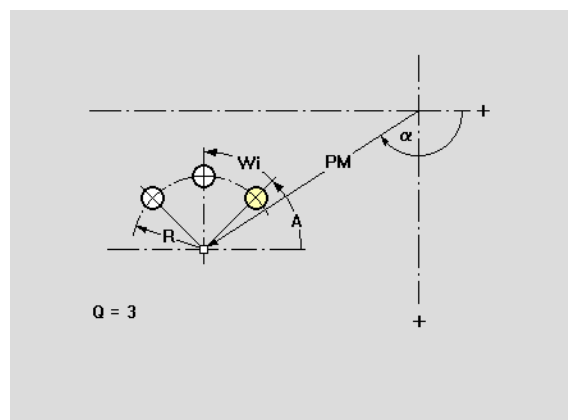
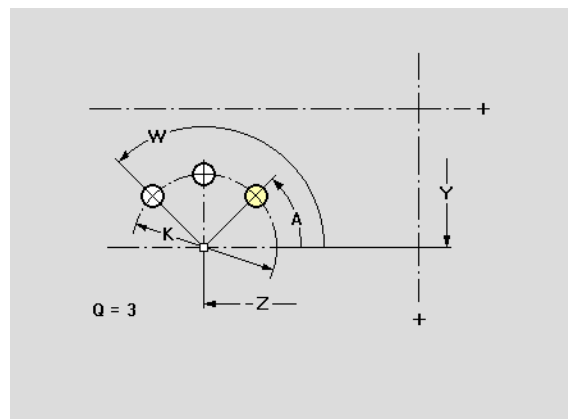
This function defines a circular drilling pattern in the YZ plane.

Parameters

- Y Center of pattern in Cartesian coordinates
- Z Center of pattern in Cartesian coordinates
- a Center of pattern in polar coordinates (reference angle: positive Z axis)
- PM Center of pattern in polar coordinates
- Q Number of holes/figures
- Orientation:
- Clockwise
 - Counterclockwise
- R Radius of the pattern
- K Diameter of the pattern
- A Starting angle, position of the first hole (reference: Z axis)
- W End angle, position of the last hole (reference: Z axis)
- Wi Angle between two holes (algebraic sign has no effect)
- Description of the hole (see "XY plane: Single hole" on page 77)

Special cases of starting and end angles (A, W):

- Without A and W: Figures are arranged on a full circle, starting at 0°
- Without W: Figures are arranged on a full circle

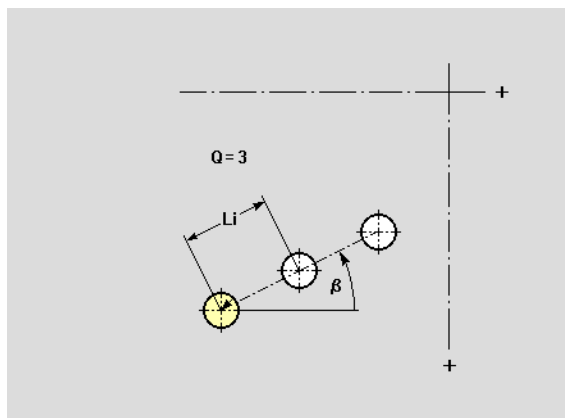
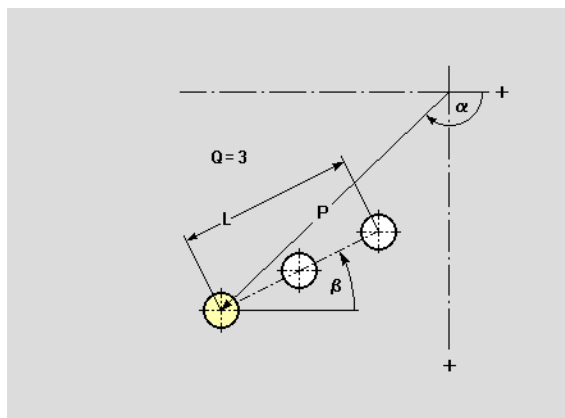
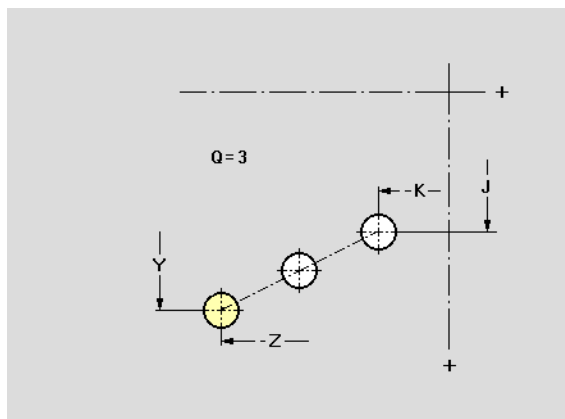


YZ plane: Linear figure pattern

This function defines a linear figure pattern in the YZ plane.

Parameters

Y	Starting point of pattern in Cartesian coordinates
Z	Starting point of pattern in Cartesian coordinates
a	Starting point of pattern in polar coordinates (reference angle: positive Z axis)
P	Starting point of pattern in polar coordinates
Q	Number of figures
J	End point of pattern in Cartesian coordinates
K	End point of pattern in Cartesian coordinates
Ji	Distance in Y direction between two figures
Ki	Distance in Z direction between two figures
b	Angle to longitudinal axis of pattern (reference: Z axis)
L	Total length of pattern
Li	Distance between two figures (pattern distance)
	Description of the figure



YZ plane: Circular figure pattern

This function defines a circular figure pattern in the YZ plane.

Parameters

- Y Center of pattern in Cartesian coordinates
- Z Center of pattern in Cartesian coordinates
- a Center of pattern in polar coordinates (reference angle: positive Z axis)
- PM Center of pattern in polar coordinates
- Q Number of figures
- Orientation:
- Clockwise
 - Counterclockwise
- R Radius of the pattern
- K Diameter of the pattern
- A Starting angle, position of the first figure (reference: Z axis)
- W End angle, position of the last figure (reference: Z axis)
- Wi Angle between two figures (algebraic sign has no effect)
- Position of the figures
- Normal position: The original figure is rotated about the center of the pattern (rotation)
 - Original position: The position of the original figure is maintained (translation)

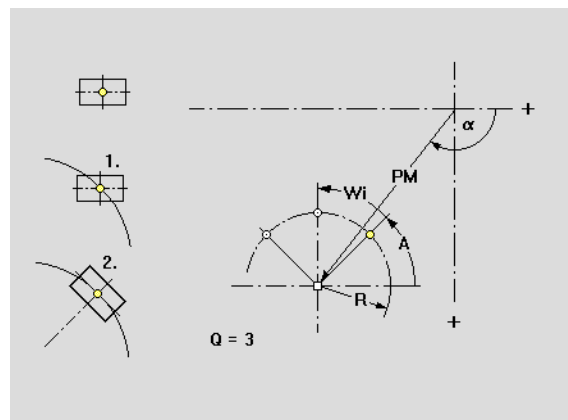
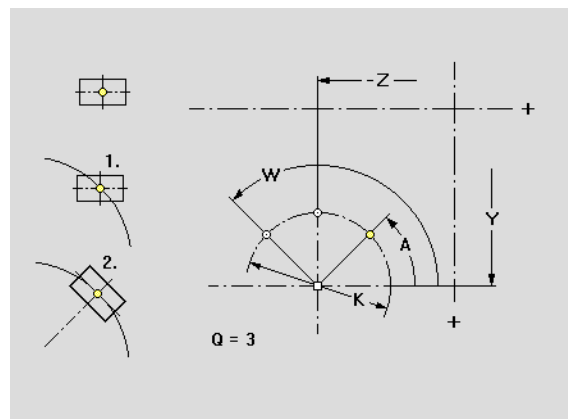
Description of the figure

Special cases of starting and end angles (A, W):

- Without A and W: Figures are arranged on a full circle, starting at 0°
- Without W: Figures are arranged on a full circle



When defining patterns with circular slots, the "center of curvature" is added to the pattern position.

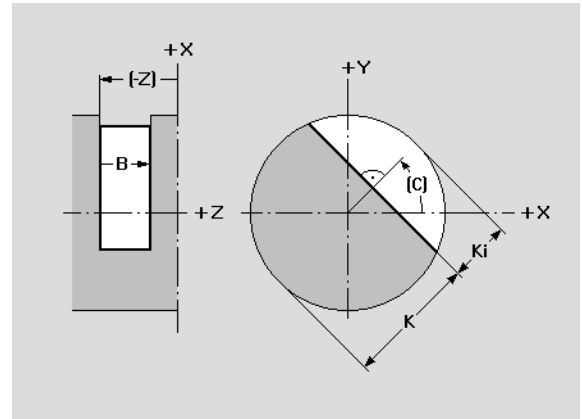


Single surface in YZ plane

This function defines a single surface in the YZ plane.

Parameters

- Ki Depth (of material to be milled)
- K Residual depth (remaining material)
- B Width (reference: reference edge Z)
 - $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction

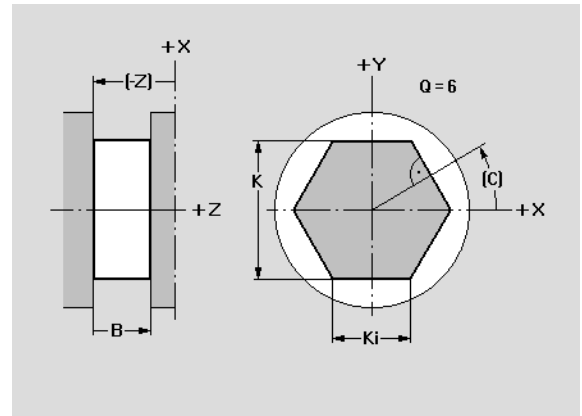


Centric polygons in YZ plane

This function defines polygonal surfaces in the YZ plane.

Parameters

- Q Number of sides ($Q \geq 2$)
- K Width across flats (inscribed circle diameter)
- Ki Edge length
- B Width (reference: reference edge Z)
 - $B < 0$: Surface in negative Z direction
 - $B > 0$: Surface in positive Z direction

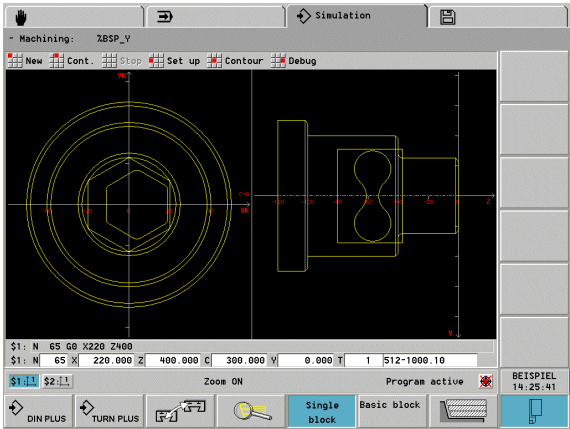


1.17 Example Programs

Machining with the Y axis

The following NC program mills first the “polygonal surface” (front face) and then a single surface. A “figure eight”-shaped pocket is then milled into the single surface.

Example: “Y axis [BSP_Y.NC]”



PROGRAMMKOPF [PROGRAM HEAD]	
#DATUM	01.03.07 [DATE]
#MATERIAL	ST 60-2 [MATERIAL]
#SCHLITTEN	\$1 [SLIDE]
#SYNCHRO	0
REVOLVER 1 [TURRET]	
T 1	ID"512-1000.10"
T 2	ID"111-80-080.1"
T 3	ID"521-1400.10"
T 4	ID"121-55-040.1"
T 5	ID"511-1000.10"
ROHTEIL [BLANK]	
N 1	G20 X100 Z150 K1
FERTIGTEIL [FINISHED PART]	
N 2	G0 X0 Z-120
N 3	G1 Z0
N 4	G1 X50 B-2
N 5	G1 Z-40 B3
N 6	G1 X80 B-2
N 7	G1 Z-100
N 8	G1 X100 B-2

N 9 G1 Z-120	
N 10 G1 X0	
STIRN_Y Z0 X50 C0 [FRONT_Y]	[Define front face of centric polygon]
N 11 G308	
N 12 G477 Z0 K40 B-3 I50 C0 Q6	
N 13 G309	
STIRN_Y Z0 X46 C0 [FRONT_Y]	[Define front face of polygon]
N 14 G308 P-2	
N 15 G377 X4 Y0 K-30 A30 R3 Q6	
N 16 G309	
MANTEL_Y X80 C90 [SURFACE_Y]	
N 17 G308	[Define single surface]
N 18 G386 Z-37 KI15 B-43 X80 C90	
N 19 G308 P-2	[Define "figure eight"-shaped pocket]
N 20 G180 Z-53 Y0	
N 21 G181 Y? B12 Q1	
N 22 G183 Z-61 YI0 R12 K-57 J-18 B12	
N 23 G181 Y? A-90 B12	
N 24 G183 Z-53 YI0 R12 K-57 J18 Q1 B12	
N 25 G181 Y0	
N 26 G309	
N 27 G309	
BEARBEITUNG [MACHINING]	
N 28 G0 Y0	[// Roughing - transverse - outside - front face]
N 29 G701 X380 Z500	
N 30 G26 S4000	
N 31 T2	
N 32 G96 S150 G95 F0.3 M4	
N 33 G0 X106 Z4	
N 34 G47 P3	
N 35 G820 NS4 NE4 P1 I1 K0.3 E0 Z-134 A90 W270 Q2 V3 D4	
N 36 G0 X52	
N 37 G0 Z4	
N 38 G95 F0.5	[// Roughing - longitudinal - outside]

1.17 Example Programs

N 9 G1 Z-120	
N 10 G1 X0	
STIRN_Y Z0 X50 C0 [FRONT_Y]	[Define front face of centric polygon]
N 11 G308	
N 12 G477 Z0 K40 B-3 I50 C0 Q6	
N 13 G309	
STIRN_Y Z0 X46 C0 [FRONT_Y]	[Define front face of polygon]
N 14 G308 P-2	
N 15 G377 X4 Y0 K-30 A30 R3 Q6	
N 16 G309	
MANTEL_Y X80 C90 [SURFACE_Y]	
N 17 G308	[Define single surface]
N 18 G386 Z-37 KI15 B-43 X80 C90	
N 19 G308 P-2	[Define "figure eight"-shaped pocket]
N 20 G180 Z-53 Y0	
N 21 G181 Y? B12 Q1	
N 22 G183 Z-61 YI0 R12 K-57 J-18 B12	
N 23 G181 Y? A-90 B12	
N 24 G183 Z-53 YI0 R12 K-57 J18 Q1 B12	
N 25 G181 Y0	
N 26 G309	
N 27 G309	
BEARBEITUNG [MACHINING]	
N 28 G0 Y0	[// Roughing - transverse - outside - front face]
N 29 G701 X380 Z500	
N 30 G26 S4000	
N 31 T2	
N 32 G96 S150 G95 F0.3 M4	
N 33 G0 X106 Z4	
N 34 G47 P3	
N 35 G820 NS4 NE4 P1 I1 K0.3 E0 Z-134 A90 W270 Q2 V3 D4	
N 36 G0 X52	
N 37 G0 Z4	
N 38 G95 F0.5	[// Roughing - longitudinal - outside]

N 39 G0 X106 Z3.3	
N 40 G47 P3	
N 41 G810 NS5 NE9 P1 I1 K0.3 E0 Z-134 A0 W180 Q2 V1 D4	
N 42 G0 Z3.3	
N 43 G0 X106	
N 44 G0 X210 Z465	
N 45 T4	[// Finishing - transverse - outside - front face]
N 46 G96 S200 G95 F0.25 M4	
N 47 G0 X52 Z3	
N 48 G47 P2	
N 49 G890 NS4 NE4 V3 H3 D3	
N 50 G47 P2	[// Finishing - contour parallel - outside]
N 51 G890 NS5 NE9 V1 H0 D1 I106 K-117	
N 52 G0 X210 Z464	
N 53 G126 S4000	[// Milling - surface 10mm - outside - front face]
N 54 M5	
N 55 T1	
N 56 G17	
N 57 G197 S637 G193 F0.1 M103	
N 58 M14	
N 59 G0 X64 Z3	
N 60 G0 Y0	
N 61 G147 I2 K2	
N 62 G843 NS12 P1 U0.5 V0.5	
N 63 G0 X64 Z3	
N 64 G0 Y0	
N 65 G0 X220 Z400	
N 66 M105	
N 67 T3	[// Milling - surface 14mm - outside - lateral surface]
N 68 G19	
N 69 G197 S455 G193 F0.1 M103	
N 70 G0 X106 Z-37	
N 71 G0 Y0	
N 72 G147 I2 K2	
N 73 G841 NS18 P1 V0.5	
N 74 G0 X106 Z-37	
N 75 G0 Y0	
N 76 G0 X180 Z500	

1.17 Example Programs

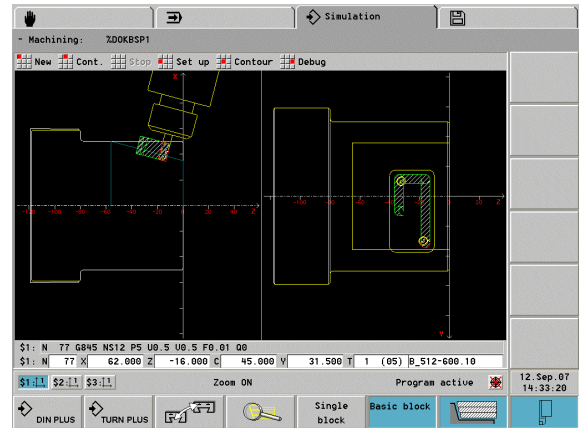
N 77 M105	
N 78 T1	[// Milling - contour 10mm - outside - front face]
N 79 G17	
N 80 G197 S637 G193 F0.1 M103	
N 81 G0 X8 Z3	
N 82 G0 Y0	
N 83 G147 I2 K2	
N 84 G845 NS15 P3 I1 U0.5 V0.5 H1 Q0	
N 85 G0 X8 Z3	
N 86 G0 Y0	
N 87 G0 X220 Z400	
N 88 M105	
N 89 T5	[// Milling - free figure - outside - lateral surface]
N 90 G19	
N 91 G197 S637 G193 F0.1 M103	
N 92 G0 X106 Z-50	
N 93 G0 Y11	
N 94 G147 I2 K2	
N 95 G845 NS23 P1 U0.5 V0.5 H1 Q0	
N 96 G0 X106 Z-50	
N 97 G0 Y0	
N 98 G0 X140 Z500	
N 99 M105	
N 100 G0 Y0	
N 101 G18	
N 102 M15	
N 103 M30	
ENDE [END]	

Machining with the B axis

The following NC program mills first a “surface” (tilted plane) and then a pocket in the tilted plane. It subsequently drills holes inside the pocket.

In this example, the tilted plane will be milled by programming single commands. In the contour definition, however, this single surface is defined. In this way, you can define the figures and holes with reference to the tilted plane. Another advantage is that the surface is displayed in the simulation.

Example: “B axis [DOKBSP1.NC]”



PROGRAMMKOPF [PROGRAM HEAD]	
#DATE	01.03.07
#MATERIAL	ST 60-2 [MATERIAL]
#SCHLITTEN	\$1 [SLIDE]
#SYNCHRO	0
SCHEIBENMAGAZIN [PLATE MAGZN.]	
ID"B_112-80-0.8"	
ID"B_512-600.10"	
ID"B_322-1000.10"	
ID"B_332-0500.10"	
ID"372-600.10"	
ID"B_522-6000.1"	
ROHTEIL [WORKPIECE BLANK]	
N 1 G20 X120 Z120 K1	
FERTIGTEIL [FINISHED PART]	
N 2 G0 X0 Z-118	
N 3 G1 Z0	
N 4 G1 X100 B-1	
N 5 G1 Z-80 B2	
N 6 G1 X118 B-1	
N 7 G1 Z-118 B-1	
N 8 G1 X0	

1.17 Example Programs

MANTEL_Y X100 C45 B75 I35 K0 [SURFACE_Y]	[Definition of the tilted plane]
N 9 G308	
N 10 G386 Z0 KI15 B-65 X100 C45	[Define surface]
N 11 G308 P-8	
N 12 G385 Z-25 Y10 A90 K55 B30 R4	[Define rectangular pocket]
N 13 G308 P-12	
N 14 G481 Z-17.5 Y30 K-32.5 J-10 Q2	[Linear drilling pattern]
N 15 G380 B5 P12 W118 I6 J8 K2 F1 V0 A90 O6	
N 16 G309	
N 17 G309	
N 18 G309	
BEARBEITUNG [MACHINING]	
N 19 G0 Y0	[// Roughing - transverse - outside - front face]
N 20 G14 Q0	
N 21 G26 S4000	
N 22 G714 ID"B_112-80-0.8" B90 O1 C0	
N 23 G96 S220 G95 F0.4 M3	
N 24 M108	
N 25 G0 X126 Z4	
N 26 G47 P3	
N 27 G820 NS4 NE4 P2.5 I1 K0.3 E0 Z-104 A90 W270 Q2 V3 D4	
N 28 G0 X104	
N 29 G0 Z4	
N 30 G14 Q0	
N 31 M109	
N 32 G95 F0.2	[// Roughing - longitudinal - outside]
N 33 M108	
N 34 G0 X126 Z3.3	
N 35 G47 P3	
N 36 G810 NS5 NE7 P5 I1 K0.3 E0 Z-104 A0 W180 Q2 V1 D4	
N 37 G0 Z3.3	
N 38 G0 X126	
N 39 G14 Q0	
N 40 M109	
N 41 G714 ID"B_122-55-0.8" B90 O1 C0	[// Finishing - contour parallel - outside]

N 42 G96 S250 G95 F0.2 M3	
N 43 M108	
N 44 G0 X6 Z3	
N 45 G47 P2	
N 46 G890 NS4 NE6 E0.2 V2 H0 D1 I124 K-78	
N 47 G14 Q0	
N 48 M109	
N 49 G126 S4000	[// Milling - surface, 75 degrees]
N 50 M5	
N 51 G714 ID"B_522-6000.1" 00 B75 C0	[Tool for milling the surface]
N 52 G19	[Activate the YZ plane]
N 53 G197 S2500 G193 F0.05 M103	
N 54 M14	
N 55 M108	
N 56 G0 X126 Z0 Y-60	
N 57 G110 C45	
N 58 M12	[Lock spindle]
N 59 G16 B75 I35 K0 U-35 W0 Q1	[Tilt the working plane]
N 60 G0 X83 Z-28 Y-60	
N 61 G1 Y50	
N 62 G1 X70	
N 63 G1 Y-60	
N 64 G0 X100	
N 65 G16 Q0	[Reset the working plane]
N 66 G0 X126 Z-25	
N 67 G0 Y0	
N 68 G14 Q0	
N 69 M105	
N 70 M109	
N 71 G714 ID"B_512-600.10" 00 B75 C0	[// Milling - pocket 6mm - outside - lateral surface]
N 72 G197 S1485 G193 F0.05 M103	
N 73 M108	
N 74 G0 X126 Z-25	
N 75 G0 Y10	
N 76 G147 I2 K2	
N 77 G845 NS12 P5 U0.5 V0.5 F0.01 Q0	[Pocket milling]
N 78 G0 X126 Z-25	

1.17 Example Programs

N 79 G0 Y0	
N 80 G14 Q0	
N 81 M105	
N 82 M109	
N 83 G714 ID"B_322-1000.10" 00 B75 C0	[// Centering 10mm - outside - lateral surface]
N 84 G197 S1146 G195 F0.1 M103	
N 85 M108	
N 86 G0 X126 Z-25	
N 87 G147 K2	
N 88 G72 NS15 K75	
N 89 G14	
N 90 M105	
N 91 M109	
N 92 G714 ID"B_332-0500.10" 00 B75 C0	[// Drilling 5mm - outside - lateral surface]
N 93 G197 S2228 G195 F0.08 M103	
N 94 M108	
N 95 G0 X126 Z-17.5	
N 96 G0 Y30	
N 97 G147 K2	
N 98 G71 NS15 E0.05 K75	
N 99 G0 X126 Z-32.5	
N 100 G0 Y0	
N 101 G14 Q0	
N 102 M105	
N 103 M109	
N 104 G714 ID"372-600.10" 00 B75 C0	[// Thread M6 - outside - lateral surface]
N 105 G197 S1000 G195 F1 M103	
N 106 M108	
N 107 G0 X126 Z-17.5	
N 108 G0 Y30	
N 109 G147 K5	
N 110 G73 NS15 B5 K75	
N 111 G0 X126 Z-32.5	
N 112 G0 Y0	
N 113 G14 Q0	
N 114 M105	
N 115 M109	

N 116 G0 Y0	
N 117 G18	
N 118 M15	
N 119 M30	
ENDE [END]	

A

Area milling

- Area milling - finishing G842 (DIN PLUS) ... 53
- Area milling - roughing G841 (DIN PLUS) ... 52

B

B axis

- Basics ... 8
- Flexible use of tools ... 9
- Multipoint tools ... 10
- Position display ... 9
- Tilting in manual control mode ... 17
- Tool orientation ... 9

C

Centric polygon

- DIN PLUS
 - XY plane G477 ... 31
 - YZ plane G487 ... 39
- TURN PLUS
 - XY plane ... 88
 - YZ plane ... 103

Circular arc

- DIN PLUS
 - Circular arc in XY contour, G172 ... 25
 - Circular arc in XY contour, G173 ... 25
 - Circular arc milling (Y axis) G12 ... 51
 - Circular arc milling (Y axis) G13 ... 51
 - Circular arc milling (Y axis) G2 ... 50
 - Circular arc milling (Y axis) G3 ... 50
- G182 (YZ plane) ... 33
- G183 (YZ plane) ... 33

Circular paths

- G12 (milling) ... 51
- G13 (milling) ... 51
- G2 (milling) ... 50
- G3 (milling) ... 50

Circular slot

- DIN PLUS
 - Circular slot in XY plane, G372/G373 ... 27
 - Circular slot in YZ plane, G382/G383 ... 35

Contour definition

- DIN PLUS
 - XY plane ... 24
 - YZ plane ... 32
- TURN PLUS
 - XY plane ... 73
 - YZ plane ... 89

Cutting limit ... 20

D

Define tool position G712 ... 47

Drilling

- TURN PLUS
 - Hole in XY plane ... 77
 - Hole in YZ plane ... 92

E

Eccentric polygon

- DIN PLUS
 - Polygon in XY plane, G377 ... 28
 - Polygon in YZ plane, G387 ... 36

Engraving

- Engrave in XY plane G803 ... 63
- Engrave in YZ plane G804 ... 64

Example

- Machining with the B axis ... 109
- Machining with the Y axis ... 104

F

FRONT_Y - Section code ... 22

Full circle

- DIN PLUS
 - Full circle in XY plane, G374 ... 27
 - Full circle in YZ plane, G384 ... 35
- G374 (XY plane) ... 27

G

G contour commands

- G472 Circular pattern (XY plane) ... 30

G functions for contour description

- G170 Starting point of contour in XY plane ... 24
- G171 Line segment in XY plane ... 24
- G172 Circular arc in XY plane ... 25
- G173 Circular arc in XY plane ... 25
- G180 Starting point of contour in YZ plane ... 32
- G181 Line segment in YZ plane ... 32

- G182 Circular arc in YZ plane ... 33
- G183 Circular arc in YZ plane ... 33
- G370 Hole in XY plane ... 26
- G371 Linear slot in XY plane ... 26
- G372 Circular slot in XY plane ... 27
- G373 Circular slot in XY plane ... 27
- G374 Full circle in XY plane ... 27
- G375 Rectangle in XY plane ... 28
- G376 Single surface in XY plane ... 31
- G377 Eccentric polygon in XY plane ... 28
- G380 Hole in YZ plane ... 34
- G381 Linear slot in YZ plane ... 34
- G382 Circular slot in YZ plane ... 35
- G383 Circular slot in YZ plane ... 35
- G384 Full circle in YZ plane ... 35
- G385 Rectangle in YZ plane ... 36
- G386 Single surface in YZ plane ... 39
- G387 Eccentric polygon in YZ plane ... 36
- G471 Linear pattern in XY plane ... 29
- G472 Circular pattern in XY plane ... 30
- G477 Centric polygon in XY plane ... 31
- G481 Linear pattern in YZ plane ... 37
- G482 Circular pattern in YZ plane ... 38
- G487 Centric polygon in YZ plane ... 39

G functions for machining

- G0 Rapid traverse (Y axis) ... 42
- G1 Linear path (Y axis) ... 49
- G12 Circular path (Y axis) ... 51
- G13 Circular path (Y axis) ... 51
- G14 Tool change position (Y axis) ... 42
- G16 Tilting the working plane ... 41
- G17 XY plane ... 40
- G18 XZ plane (turning) ... 40
- G19 YZ plane ... 40
- G2 Circular path (Y axis) ... 50
- G3 Circular path (Y axis) ... 50
- G600 Preselect tool ... 48
- G701 Rapid traverse to machine coordinates (Y axis) ... 43
- G712 Define tool position ... 47
- G714 Insert magazine tool ... 44

G800 Thread milling in XY plane ... 65
 G803 Engrave in XY plane (Y axis) ... 63
 G804 Engrave in YZ plane (Y axis) ... 64
 G806 Thread milling in YZ plane ... 66
 G808 Hobbing ... 67
 G841 Area milling - roughing (Y axis) ... 52
 G842 Area milling - finishing (Y axis) ... 53
 G843 Polygon milling - roughing (Y axis) ... 54
 G844 Polygon milling - finishing (Y axis) ... 55
 G845 Pocket milling - roughing (Y axis) ... 56
 G846 Pocket milling - finishing (Y axis) ... 61

H

Hobbing G808 ... 67
 Hole
 DIN PLUS
 Hole in XY plane, G370 ... 26
 Hole in YZ plane, G380 ... 34

L

Lateral surface
 SURFACE_Y section ... 23
 Line segment
 DIN PLUS
 Line segment in XY plane, G171 ... 24
 Line segment in YZ plane, G181 ... 32
 TURN PLUS
 Line segment in XY plane ... 75
 Line segment in YZ plane ... 90
 Linear path G1 (milling) ... 49

M

Magazine list - Basics ... 11
 Magazine tool
 Changing ... 14
 Compensating ... 18
 Compensation in automatic mode ... 19
 Declaring ... 15
 Define tool position G712 ... 47
 Insert magazine tool G714 ... 44
 Measurement ... 18
 Returning tools to the magazine ... 16
 Milling contour definition, TURN PLUS ... 72
 Milling contour position
 DIN PLUS Y axis ... 20
 Milling cycles
 DIN PLUS
 Area milling - finishing G842 (Y axis) ... 53
 Area milling - roughing G841 (Y axis) ... 52
 Engrave in XY plane G803 (Y axis) ... 63
 Engrave in YZ plane G804 (Y axis) ... 64
 Pocket milling - finishing G846 (Y axis) ... 61
 Pocket milling - roughing G845 (Y axis) ... 56
 Polygon milling - finishing G844 (Y axis) ... 55
 Polygon milling - roughing G843 (Y axis) ... 54
 Milling cycles, DIN PLUS ... 52
 Milling depth ... 20
 Multipoint tools
 Entries in magazine list ... 11
 Multipoint tools for the B axis ... 10

P

Pattern
 DIN PLUS
 Circular pattern in XY plane, G472 ... 30
 Circular pattern in YZ plane, G482 ... 38
 Linear pattern in XY plane, G471 ... 29
 Linear pattern in YZ plane, G481 ... 37
 TURN PLUS
 Circular drilling pattern in XY plane ... 85
 Circular drilling pattern in YZ plane ... 100
 Circular figure pattern in XY plane ... 87
 Circular figure pattern in YZ plane ... 102
 Linear drilling pattern in XY plane ... 84
 Linear drilling pattern in YZ plane ... 99
 Linear figure pattern in XY plane ... 86
 Linear figure pattern in YZ plane ... 101
 Pocket milling
 DIN PLUS
 Pocket milling - finishing G846 (Y axis) ... 61
 Pocket milling - roughing G845 (Y axis) ... 56
 Polygon milling - finishing G844 ... 55
 Polygon milling - roughing G843 ... 54
 Positioning commands, DIN PLUS ... 42
 Preselect tool G600 ... 48

R

- Rapid traverse
 - In machine coordinates G701 (Y axis) ... 43
 - Rapid traverse G0 (Y axis) ... 42
- REAR_SIDE_Y - Section code ... 22
- Rectangle
 - DIN PLUS
 - Rectangle in XY plane, G375 ... 28
 - Rectangle in YZ plane, G385 ... 36
- Reference data, TURN PLUS
 - Y front and back ... 73
 - Y lateral surface ... 89
- Reference plane
 - FRONT_Y, REAR_SIDE_Y section ... 22
 - SURFACE_Y section ... 23

S

- Section codes for tool magazine ... 22
- Section codes for Y axis ... 22
- Single surface
 - G376 (XY plane) ... 31
 - G386 (YZ plane) ... 39
 - XY plane, TURN PLUS ... 88
 - YZ plane, TURN PLUS ... 103
- Slot
 - DIN PLUS
 - Circular slot in XY plane, G372/G373 ... 27
 - Circular slot in YZ plane, G382/G383 ... 35
 - Linear slot in XY plane, G371 ... 26
 - Linear slot in YZ plane G381 ... 34
- Slot, linear
 - DIN PLUS
 - Linear slot in XY plane, G371 ... 26
 - Linear slot in YZ plane, G381 ... 34

- Starting point of contour
 - DIN PLUS
 - Starting point of contour in XY plane, G170 ... 24
 - Starting point of contour in YZ plane, G180 ... 32
 - TURN PLUS
 - XY plane ... 74
 - YZ plane ... 89
- SURFACE_Y - Section code ... 23

T

- Tapping
 - TURN PLUS
 - XY plane ... 77
 - YZ plane ... 92
- Thread milling in XY plane G800 ... 65
- Thread milling in YZ plane G806 ... 66
- Tilted plane - Drilling and milling ... 21
- Tilted working plane - Basics ... 8
- Tilting the working plane G16 ... 41
- Tool change position
 - Tool change position G14 (Y axis) ... 42
- Tool magazine
 - Adding tools ... 12
 - Basics ... 10
 - Removing tools ... 13

W

- Working planes, DIN PLUS ... 40

X

- XY plane G17 (front or back) ... 40
- XZ plane G18 (turning) ... 40

Y

- Y axis - Basics ... 8
- Y axis - Basics, TURN PLUS ... 71
- Y back, TURN PLUS reference data ... 73
- Y front, TURN PLUS reference data ... 73
- Y lateral surface, TURN PLUS reference data ... 89
- YZ plane G19 (side view/surface) ... 40

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 5061

E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104

E-mail: service.ms-support@heidenhain.de

TNC support ☎ +49 8669 31-3101

E-mail: service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103

E-mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102

E-mail: service.plc@heidenhain.de

Lathe controls ☎ +49 8669 31-3105

E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

